



Wind-US Results for the AIAA 2nd Propulsion Aerodynamics Workshop

This presentation contains Wind-US results presented at the 2nd Propulsion Aerodynamics Workshop. The workshop was organized by the American Institute of Aeronautics and Astronautics, Air Breathing Propulsion Systems Integration Technical Committee with the purpose of assessing the accuracy of computational fluid dynamics for air breathing propulsion applications. Attendees included representatives from government, industry, academia, and commercial software companies. Participants were encouraged to explore and discuss all aspects of the simulation process including the effects of mesh type and refinement, solver numerical schemes, and turbulence modeling.

The first set of challenge cases involved computing the thrust and discharge coefficients for a 25° conical nozzle for a range of nozzle pressure ratios between 1.4 and 7.0. Participants were also asked to simulate two cases in which the 25° conical nozzle was bifurcated by a solid plate, resulting in vortex shedding (NPR=1.6) and shifted plume shock (NPR=4.0).

A second set of nozzle cases involved computing the discharge and thrust coefficients for a convergent dual stream nozzle for a range of subsonic nozzle pressure ratios. The workshop committee also compared the plume mixing of these cases across various codes and models.

The final test case was a serpentine inlet diffuser with an outlet to inlet area ratio of 1.52 and an offset of 1.34 times the inlet diameter. Boundary layer profiles, wall static pressure, and total pressure at downstream rake locations were examined.



Wind-US Results for the AIAA 2nd Propulsion Aerodynamics Workshop

Vance Dippold, III
Lancert Foster
Mina Mankbadi
NASA Glenn Research Center

July 31-August 1, 2014



Workshop Objectives

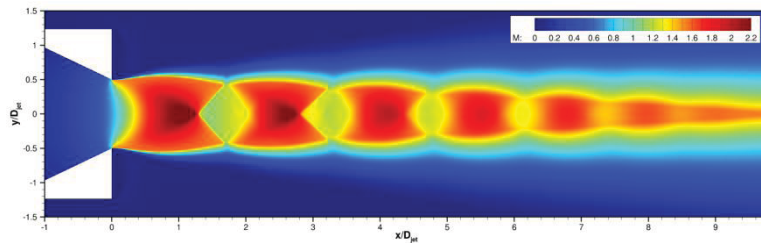
(from 2nd PAW Brochure)

- Assess the numerical prediction capability (e.g., mesh, numerics, turbulence modeling, high-performance computing requirements, and modeling techniques) of current-generation CFD technology/codes for Air Breathing Propulsion related Aerodynamic flows.
- Develop practical numerical simulation guidelines for 2-D and 3-D CFD prediction of jet related flow fields utilizing Navier-Stokes equations.
- Explore the underlying physics, flow interaction, jet mixing and dissipation flows related to Propulsion Aerodynamics.
- Enable development of more accurate prediction methods, processes, procedures and tools.
- Enhance CFD prediction capability for practical air breathing propulsion aerodynamic design and optimization.
- Provide an impartial forum for evaluating the effectiveness of existing computer codes and modeling techniques.
- Enhance interests in jet related flows and Identify areas needing additional research and development.
- Cultivate collaboration between the aerospace industry, research institutions and academia.

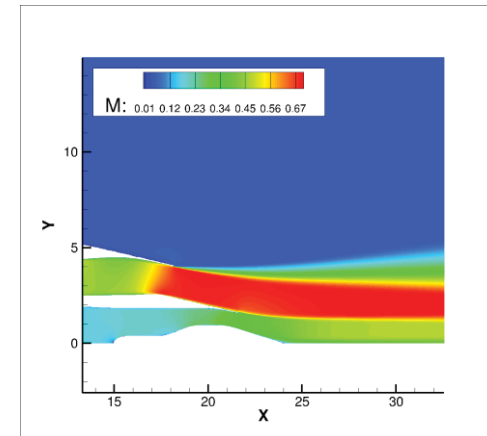


Workshop Test Cases

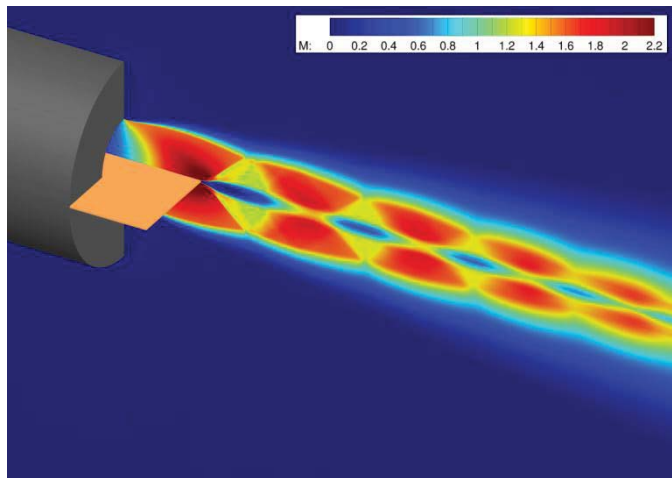
25° Conical Nozzle



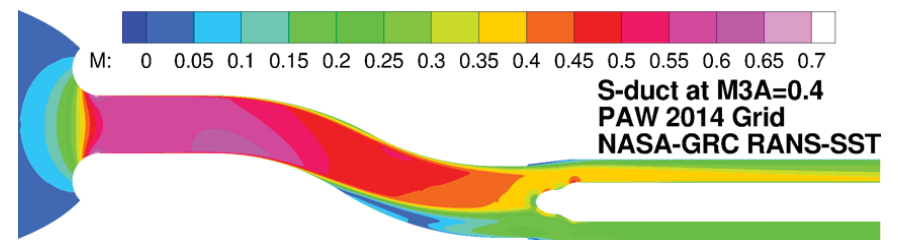
Dual Stream Nozzle



25° Conical Nozzle w/ Splitter Plate



Serpentine Inlet (S-Duct)





Workshop Format

- Two-day workshop held at the Cleveland Convention Center following the 2014 Propulsion and Energy Conference – July 31-August 1, 2014
- Each group was given 25 minutes to present each of the two sets test cases:
 - Nozzle results were presented in Room 6.
 - S-Duct results were presented in Room 7.
- Organizers will present a summary paper at 2015 Propulsion and Energy Conference.
 - Computational results will be consolidated and compared with experimental data.
 - Conclusions and observations of trends will be discussed.
- Select results will be presented at the 2015 AIAA Propulsion and Energy Conference.



References

- “An Experimental Study of Compressible Flow Through Convergent-Conical Nozzles, Including a Comparison with Theoretical Results,” R. L. Thornock & E. F. Brown, *Transactions of the ASME Journal of Basic Engineering*, pp. 926-932 , Dec. 1972.
- "A Comparison of Several CFD Codes with Experimental Data in a Diffusing S-Duct," Delot, A. and Scharnhorst, R., AIAA Paper 2013-3796, July 2013.
- “Propulsion Aerodynamic Workshop I, Comparison of Participant Analyses with Experimental Results for Convergent Conical Nozzle Flowfields and Performance,” Thornock, R. L. and Sokhey, J. S., AIAA Paper 2013-3735, July 2013.
- “Computation Simulations of Convergent Nozzle for the AIAA 1st Propulsion Aerodynamics Workshop,” Dippold, V. F., NASA/TM-2014-218329, 2014.
- Towne, C. E., “Wind-US Users Guide, Version 2.0,” NASA/TM-2009-215804, Oct. 2009.
- <http://www.grc.nasa.gov/WWW/winddocs/index.html>



Wind-US Conical Nozzle Results for the AIAA 2nd Propulsion Aerodynamics Workshop

Vance Dippold, III
Lancert Foster
Mina Mankbadi
NASA Glenn Research Center

July 31-August 1, 2014



25° Conical Nozzle

✓ Instance 1

- Axisymmetric Conical Nozzle
 - 3.0 in diameter
 - 25° half-angle
- NPR=1.4-7.0 (11 cases)
- Requested data:
 - C_d, C_v
 - M_{wall} on nozzle wall
 - M_{wall} from rake in jet plume

✓ Instance 2

- Compare jet plume for NPR=4.0:
 - 25° conical, axisymmetric
 - 25° conical w/Splitter plate
- Requested data: Flowfield p, T, M, θ

✓ Instance 3

- Time-accurate simulation of splitter plate vortex shedding for NPR=1.6
 - 25° conical w/Splitter plate
- Requested data: flowfield snap-shot

**PROPULSION AERODYNAMICS WORKSHOP
NOZZLE TEST CASES**

Instance 1
2D Axisymmetric Determination of Nozzle Coefficients, C_v , C_d

$$C_v \equiv \frac{\int_{r_0}^{r_{exit}} [\rho U^2 + (P - P_{amb})] r dr}{V_{ideal} \int_0^{r_{exit}} \rho U r dr}$$

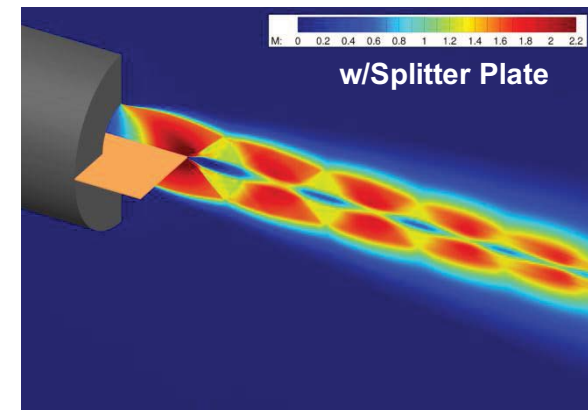
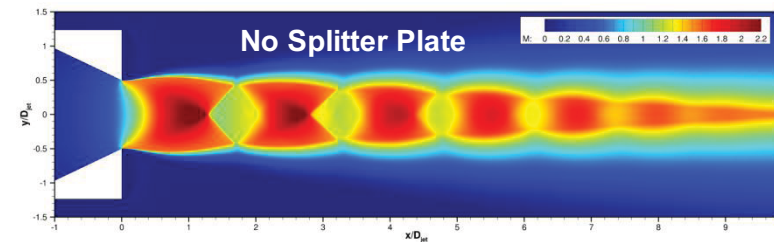
$$C_d \equiv \frac{2\pi \int_0^{r_{exit}} \rho U r dr}{\rho_{ideal} V_{ideal} A_{exit}}$$

Expressions are for integration in nozzle exit plane, however integration along any surface or, for C_v , axial force summation on flow boundaries is OK.

ρ, U, P refer to local density, axial velocity, and pressure, respectively on integration path; the subscript, "ideal", refers to quantities calculated based on average stagnation conditions and ambient pressure, P_{amb} .

		2D Test Cases											
		Nozzle Pressure Ratio, P_t/P_{amb}											
Nozzle Configuration	Reference	1.4	1.6	1.8	2	2.5	3	3.5	4	5	6	7	
	15	*	*	*	*	*	*	*	*	*	*	*	*
	25	*	*	*	*	*	*	*	*	*	*	*	*
	40	*	*	*	*	*	*	*	*	*	*	*	*

The jet boundary shape and location of the first shock can be compared with spark shadowgraphs. Preferably, all cases to be completed; at a minimum, the reference and a complete range for one half angle





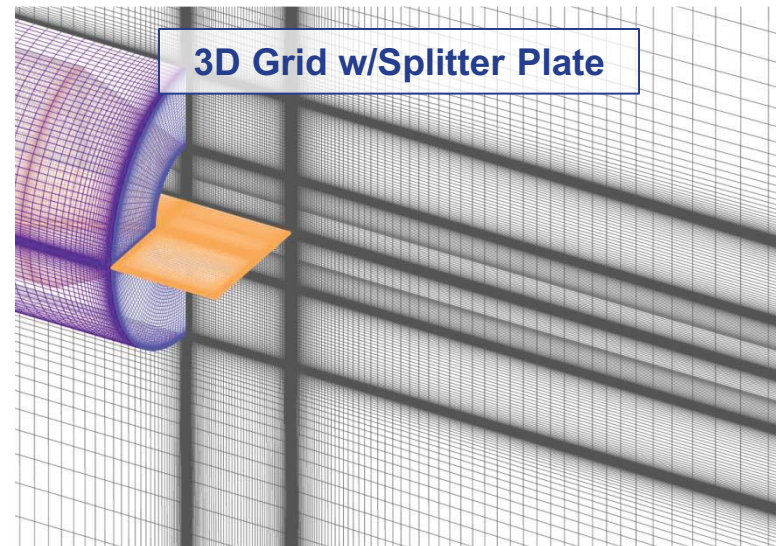
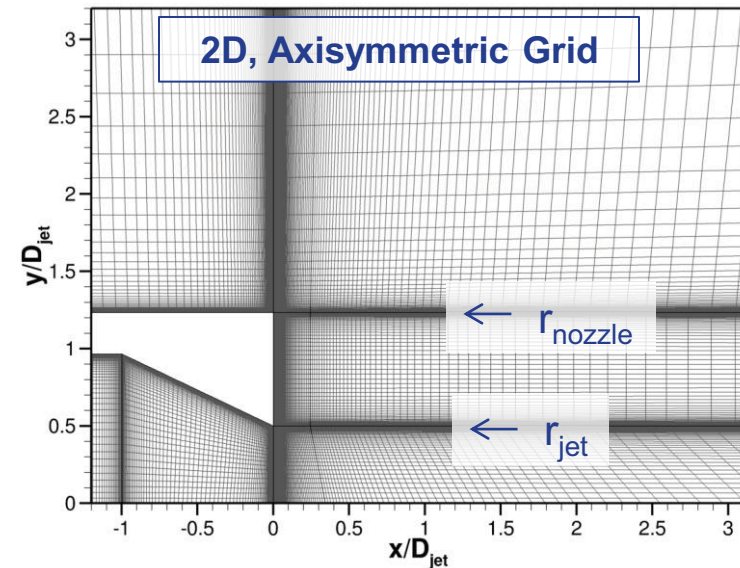
Wind-US Solver

- Wind-US v3
 - RANS and Hybrid RANS/LES solvers
 - Structured and unstructured grids
 - Numerous turbulence models, numerical schemes, and boundary conditions
- All cases:
 - Structured grid solver
 - Roe 2nd-order physical spatial integration scheme (default)
 - Minmod TVD grid flux limiter (default)
 - Inflow: p_0 , T_0 held
 - Outflow: p_{inf} held
- Axisymmetric
 - RANS with SST turbulence model (no compressibility corrections)
 - Nominal: CFL#=0.10 on fine grid
 - $\Delta t=2.0e-8$ s on fine grid for simulations showing signs of unsteadiness
- 3D w/Splitter plate, NPR=1.6
 - Spalart Detached Eddy Simulation (DES) method
 - $\Delta t=2.0e-8$ s on fine grid
 - DQ limiter on for jet plume zones
 - Zone coupling mode: average
- 3D w/Splitter plate, NPR=4.0
 - RANS with SST turbulence model (no compressibility corrections)
 - CFL#=0.10 on fine grid
 - DQ limiter on for jet plume zones
 - Zone coupling mode: average



PAW-Provided Grids

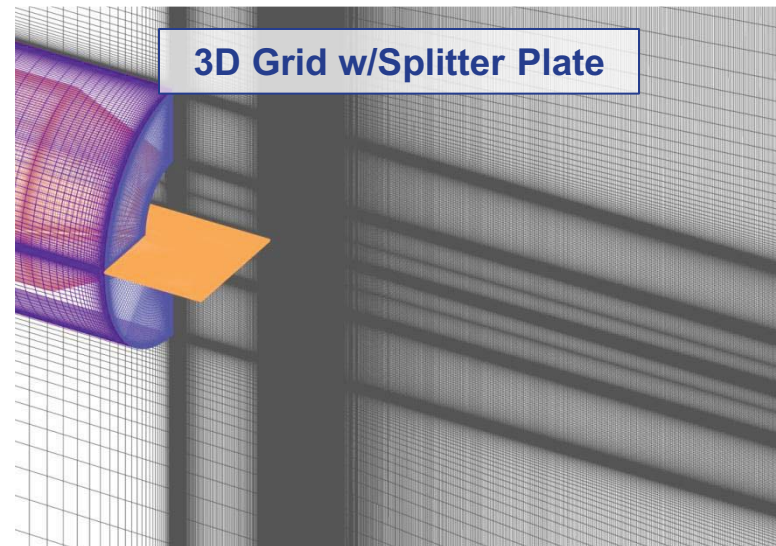
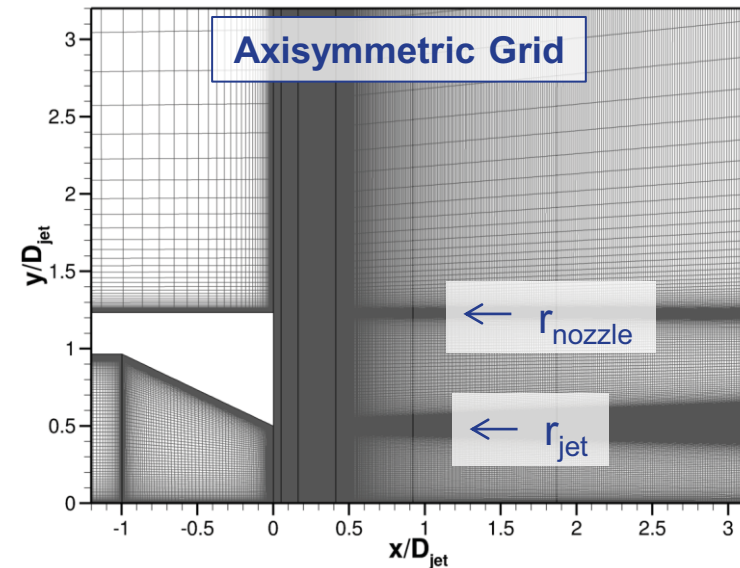
- Structured
- Radial grid lines NOT perpendicular to centerline in plume region
- $(\Delta s)_{wall} = 1e-4$ inches
- Zonal boundaries through shear layer
- Zones repartitioned for parallel processing
- Axisymmetric, Conical:
 - 71,424 cells
 - 11 zones
- 3D w/Splitter Plate:
 - Original grid: symmetry plane aligned with splitter plate – NOT favorable for vortex-shedding simulation
 - 12,719,400 cells
 - 69 zones





GRC-Developed Grids

- Created with Pointwise
- Structured
- Point-matched zonal interfaces
- Radial grids lines perpendicular to centerline
- $(\Delta s)_{wall} = 1e-5$ inches
 - for $y^+ \sim 1$ at nozzle exit
- Tighter grid spacing in jet plume:
 - Axisymmetric: $\Delta x < 0.05$ inches to resolve shock and expansion waves
 - 3D: $\Delta x < 0.005$ inches to resolve vortex structures
- Axisymmetric, w/o Splitter Plate:
 - 352,800 cells
 - 11 zones
- 3D w/Splitter Plate:
 - 104,112,000 cells
 - 262 zones



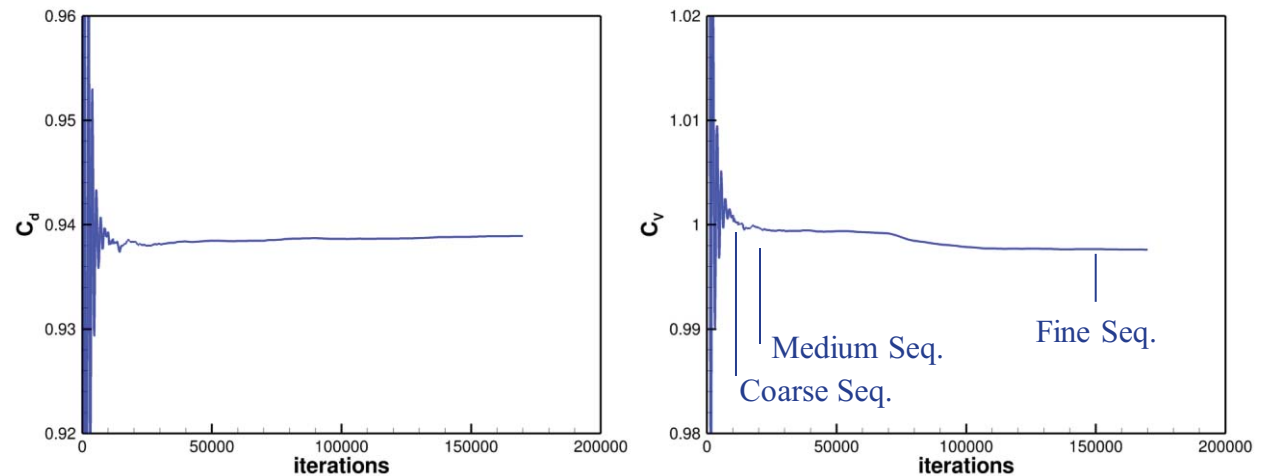


Convergence

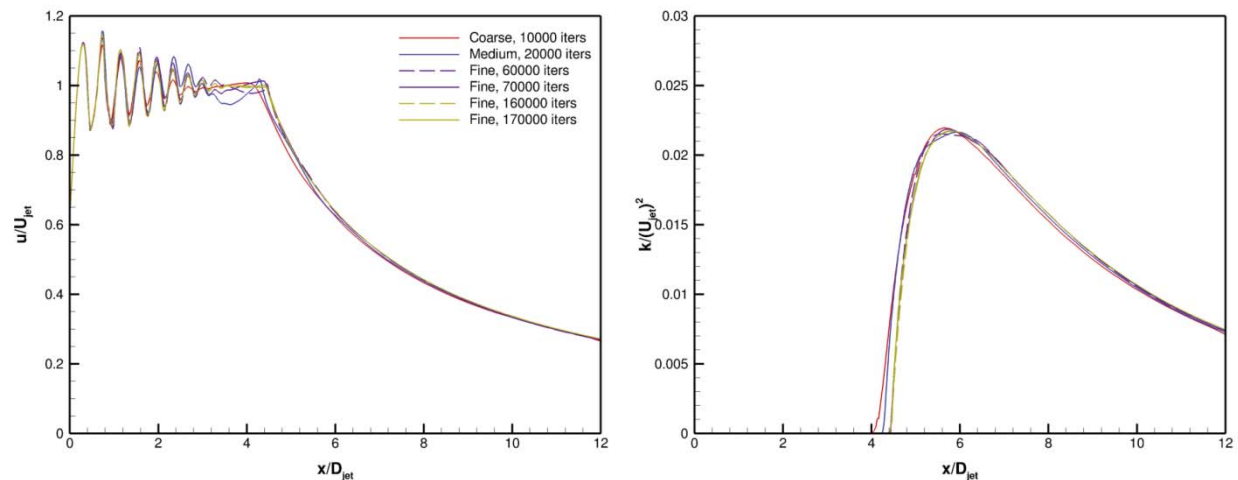
- Sequence grid:
 - Coarse: every 4th point
 - Medium: every 2nd point
 - Fine: all points
- Axisymmetric, constant CFL#:
 - 70,000-120,000 iterations
- Axisymmetric, constant Δt :
 - 300,000-400,000 iterations
- 3D, splitter plate, constant CFL#:
 - 250,000 iterations
- 3D, splitter plate, constant Δt :
 - DES: 400,000 iterations

Shown: 25° conical nozzle,
GRC grid, NPR=2.5.
Centerline u and TKE took
longer to converge than C_d
and C_v .

Nozzle Exit: C_d and C_v

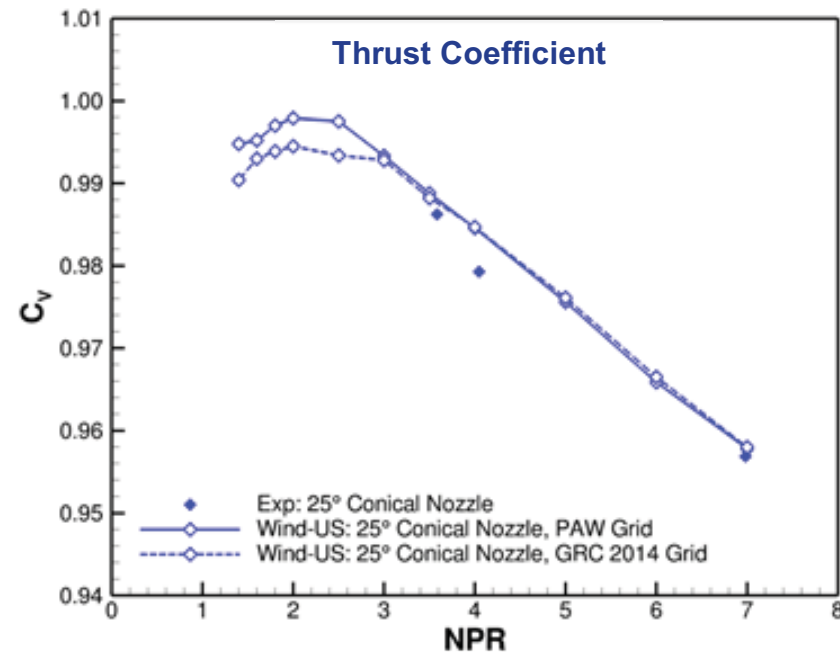
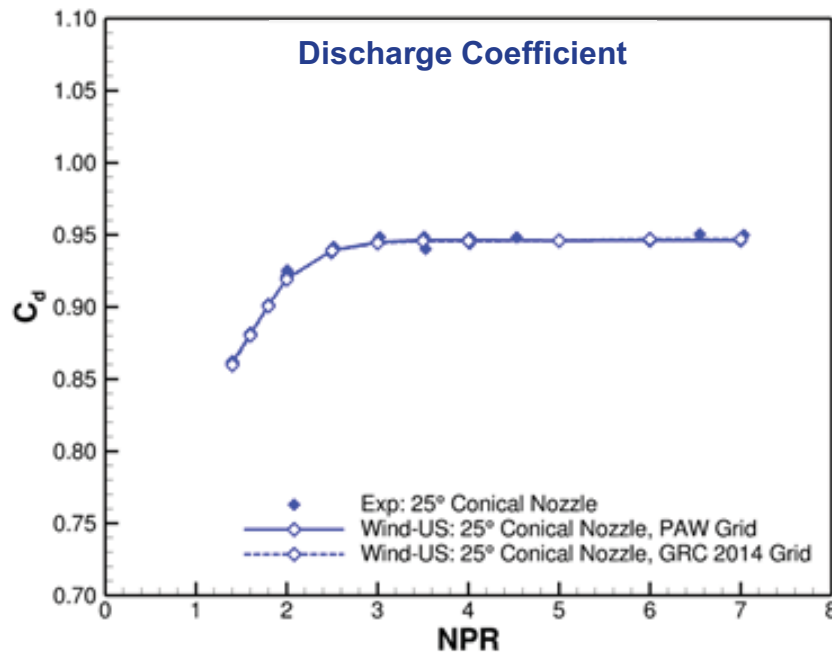


Jet Plume: Centerline u and TKE





Instance 1: Discharge & Thrust Coefficients



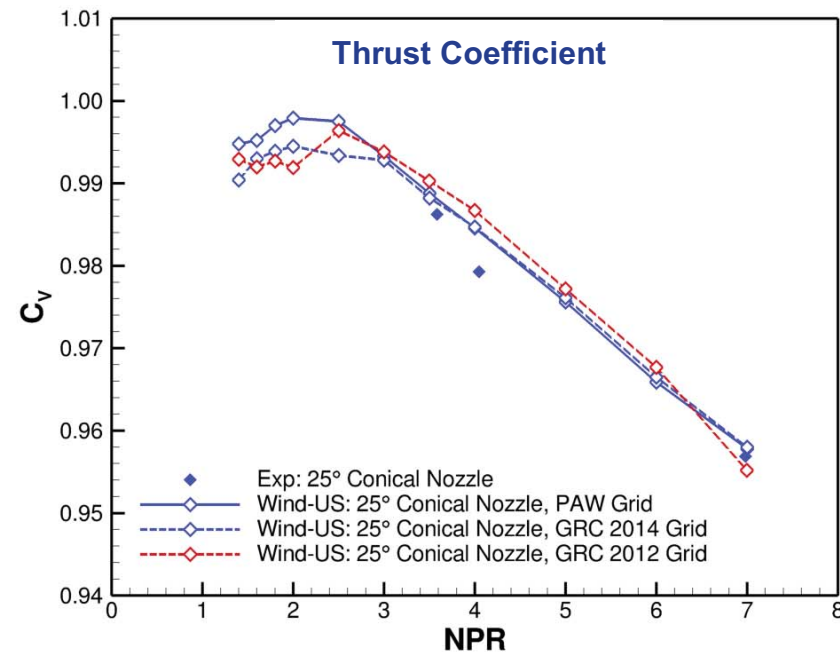
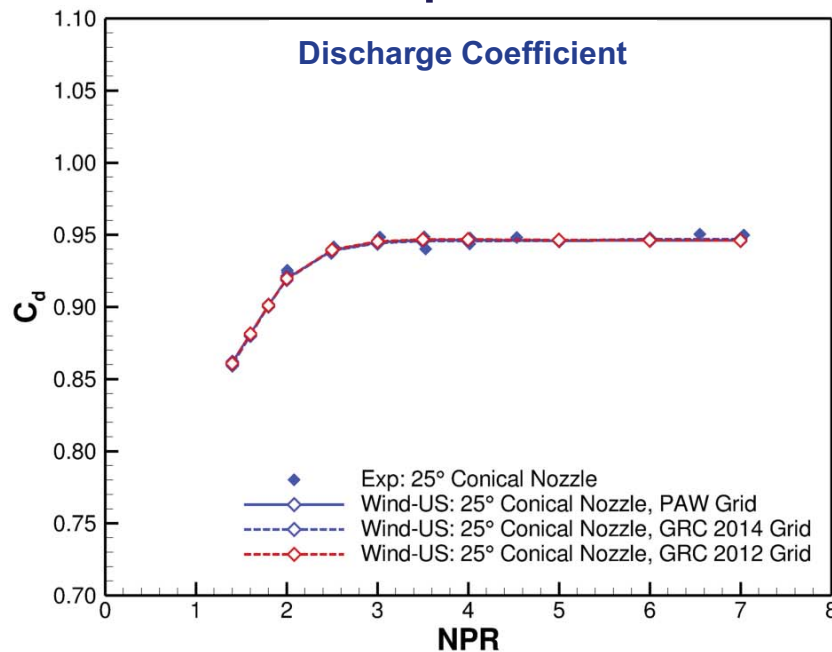
$$C_d \equiv \frac{2\pi \cdot \int_0^{r_{nozzle}} \rho \cdot u \cdot r \cdot dr}{\rho_{ideal} \cdot U_{ideal} \cdot A_{exit}}$$

$$C_V \equiv \frac{\int_0^{r_{nozzle}} [\rho \cdot u^2 + (p - p_\infty)] \cdot r \cdot dr}{U_{ideal} \cdot \int_0^{r_{nozzle}} \rho \cdot u \cdot r \cdot dr}$$

- Integrated over entire nozzle, including base region
- C_d
 - Excellent agreement between two CFD grids and experimental data for all NPRs
- C_V
 - Excellent agreement for $NPR \geq 3.0$
 - Discrepancy between CFD solutions for $NPR < 3.0$; no experimental data points for clarity



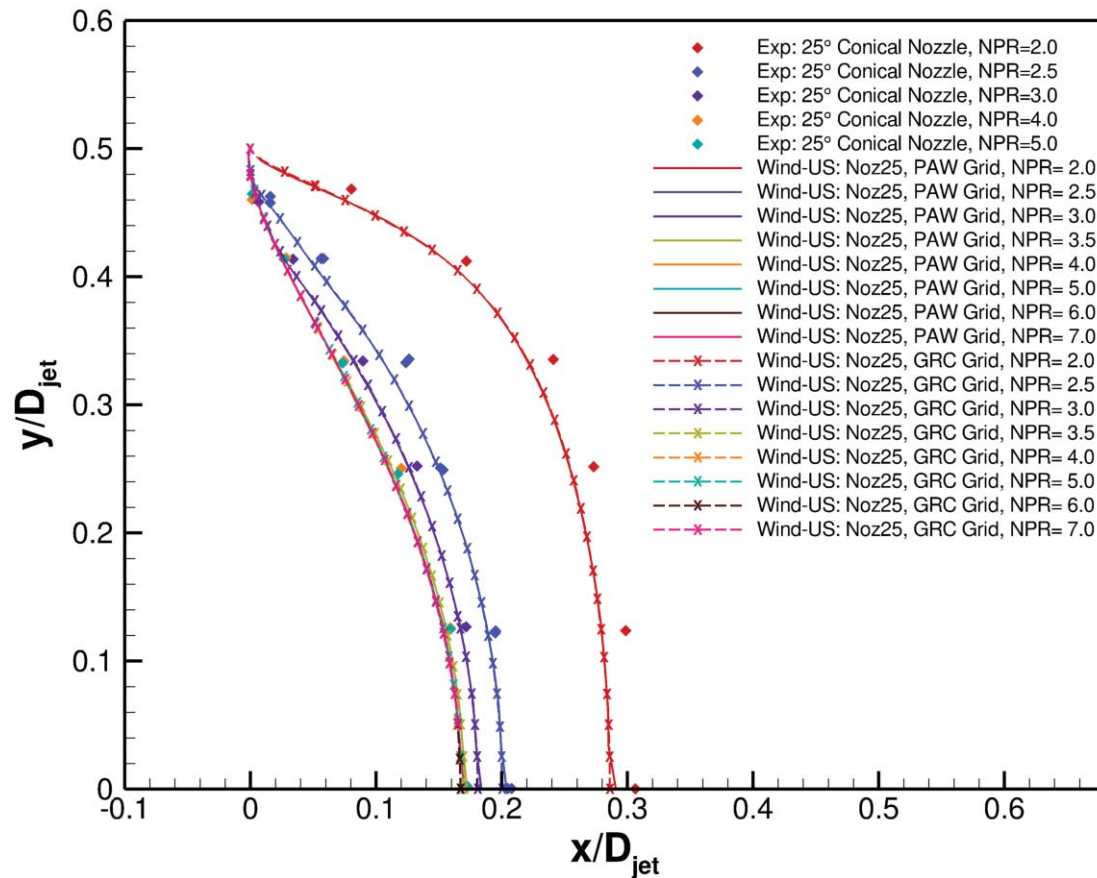
Instance 1: Discharge & Thrust Coefficients Comparison with PAW 2012 Results



- No experimental C_v dat for $NPR < 3.0$
- Compared with PAW 2012 results (71,466-cell grid)
- Still unclear which grid captures C_v for $NPR < 3.0$



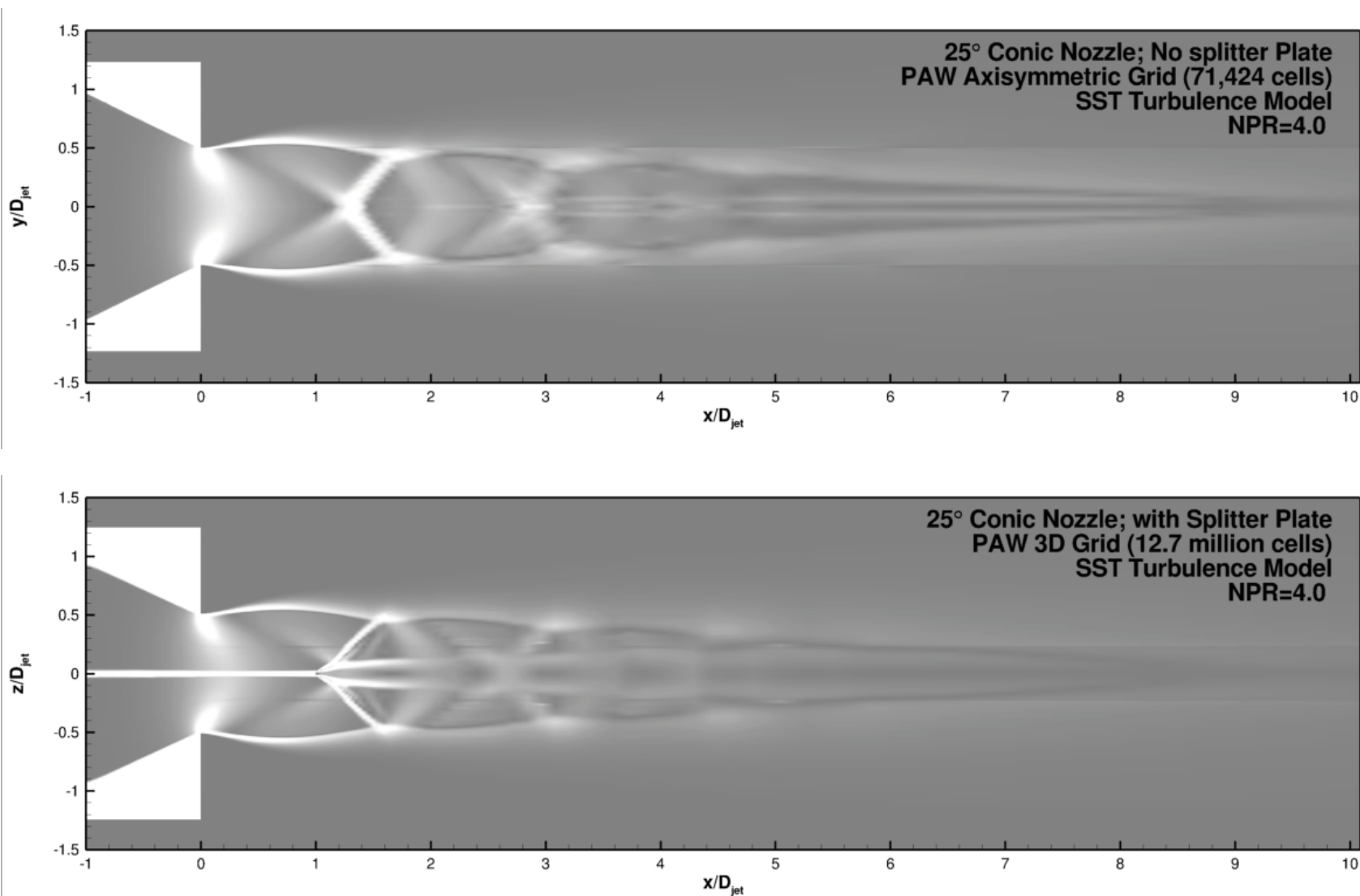
Instance 1: Jet Plume Sonic Lines (Mach=1)



- Excellent agreement between solutions of two CFD grids
- Both grids show excellent agreement with experimental data for $\text{NPR} \geq 2.5$

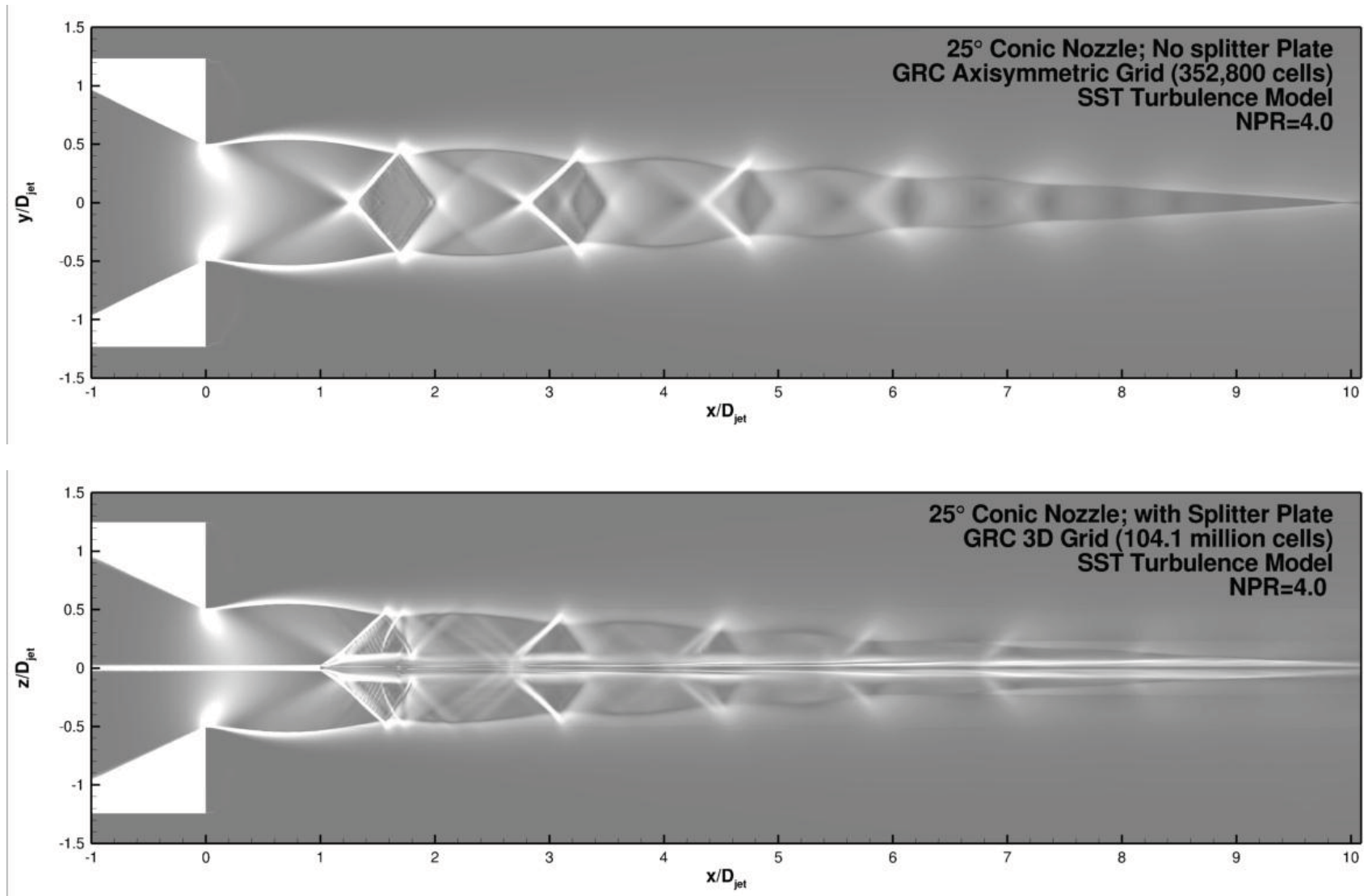


Instance 2: PAW Grids





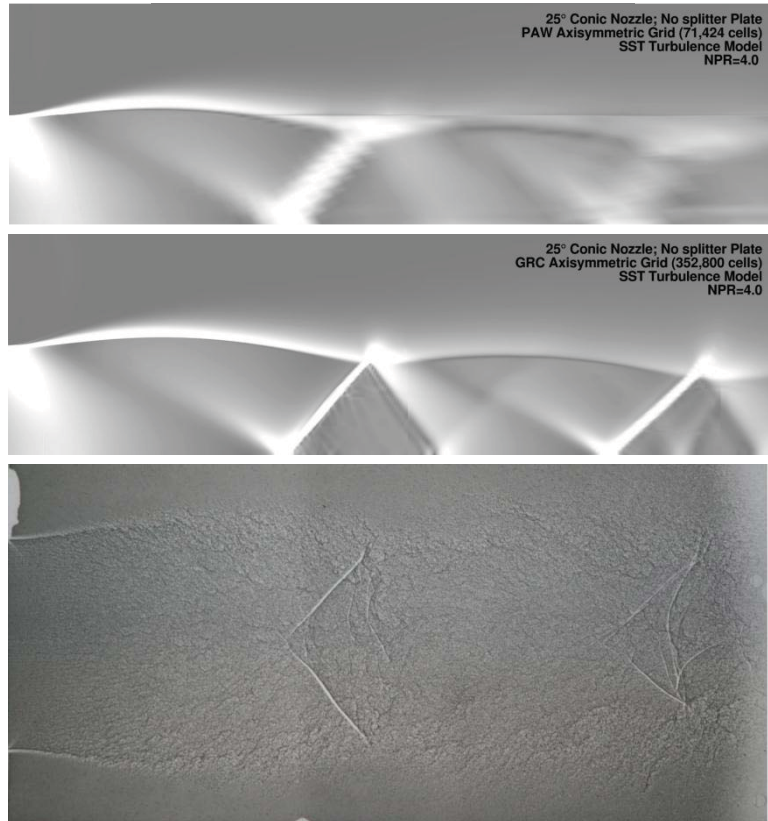
Instance 2: GRC Grids



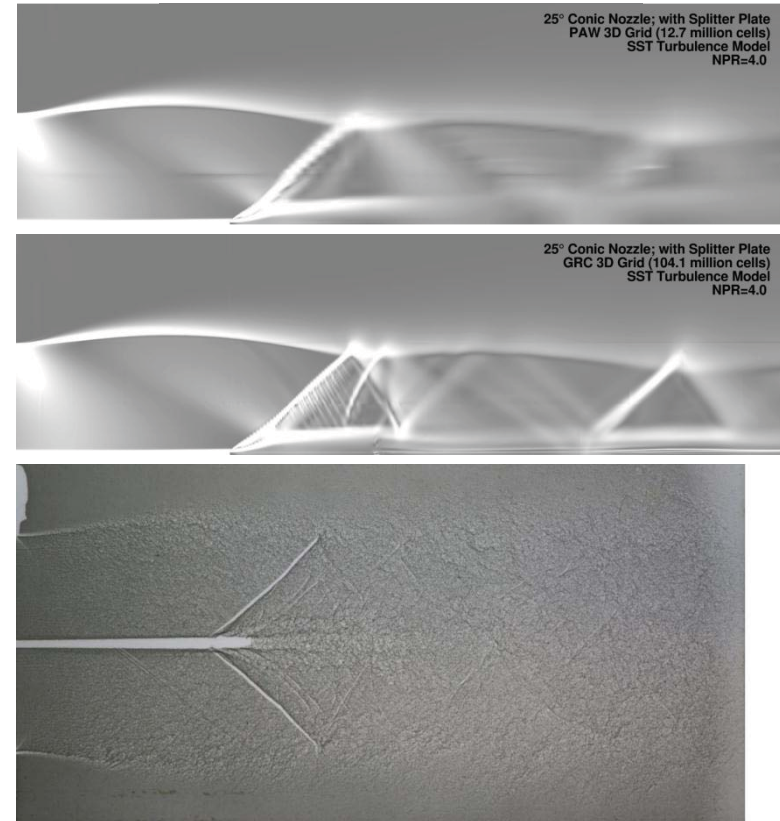


Instance 2: Comparison of Wind-US and Experimental Shadowgraphs

NPR=4.0; No Splitter Plate



NPR=4.0; With Splitter Plate

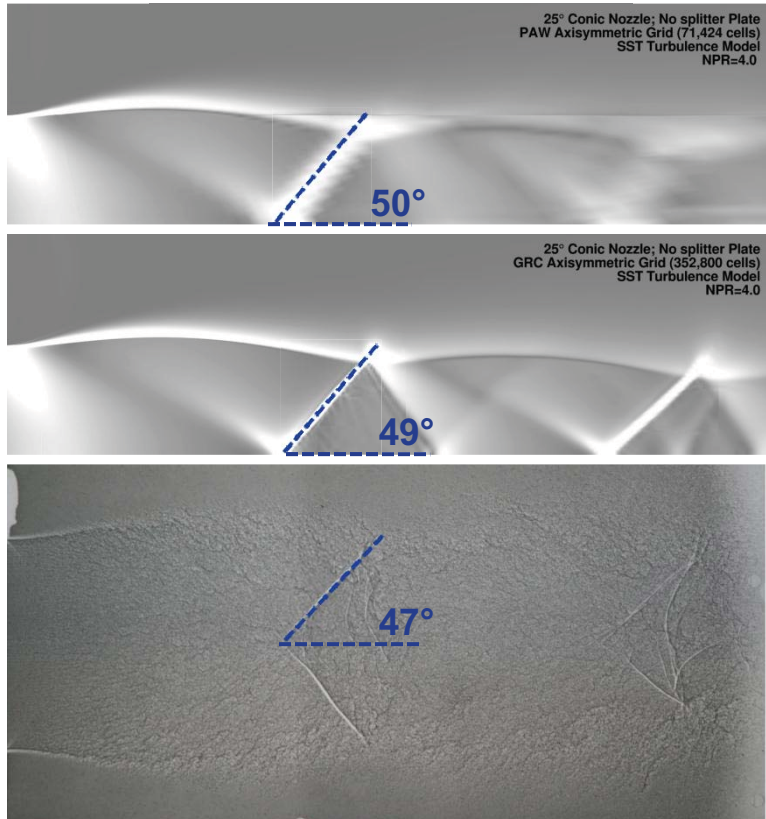


- Wind-US predicts correct locations of shock and expansion waves.
- The finer grids do a better job resolving shock and expansion waves.

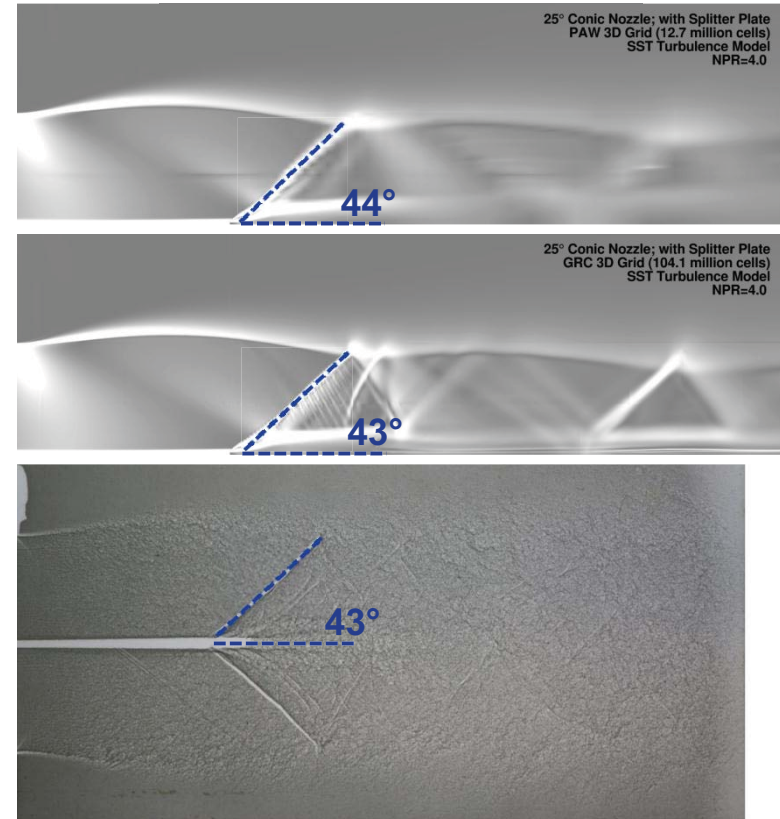


Instance 2: Comparison of Wind-US and Experimental Shadowgraphs

NPR=4.0; No Splitter Plate



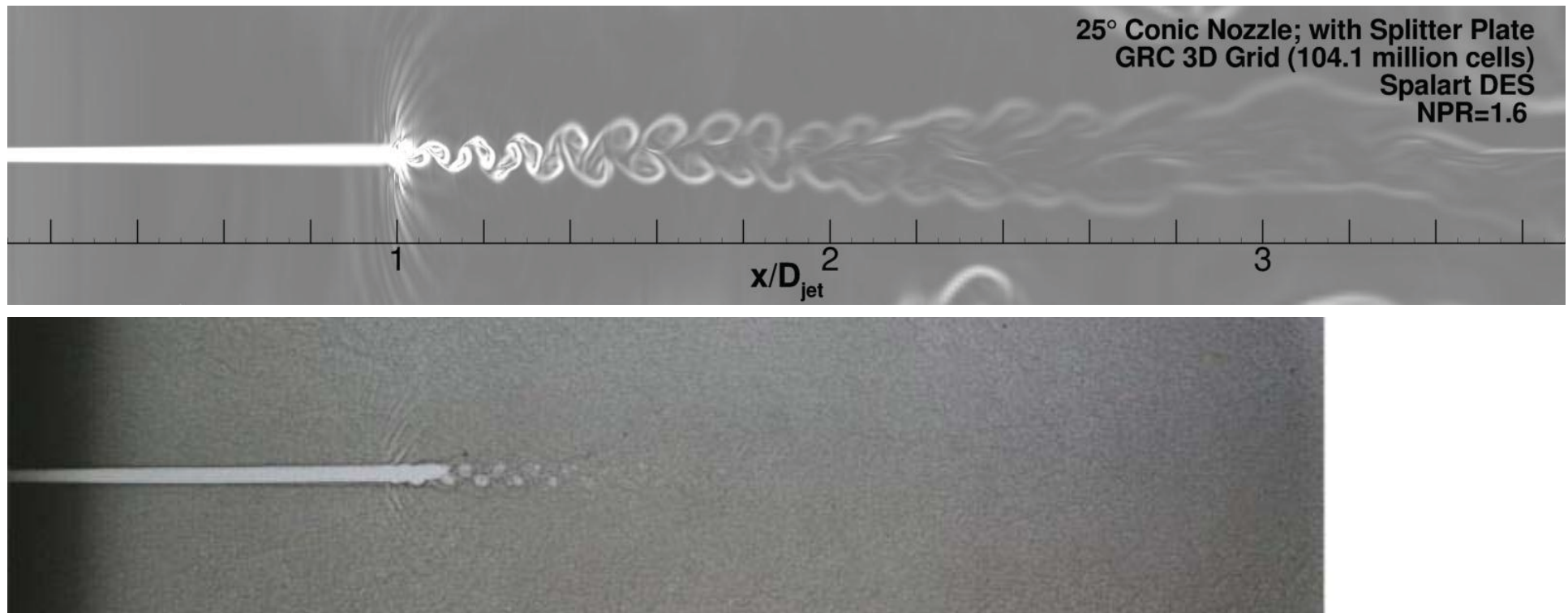
NPR=4.0; With Splitter Plate



- Wind-US predicts correct locations of shock and expansion waves.
- The finer grids do a better job resolving shock and expansion waves.



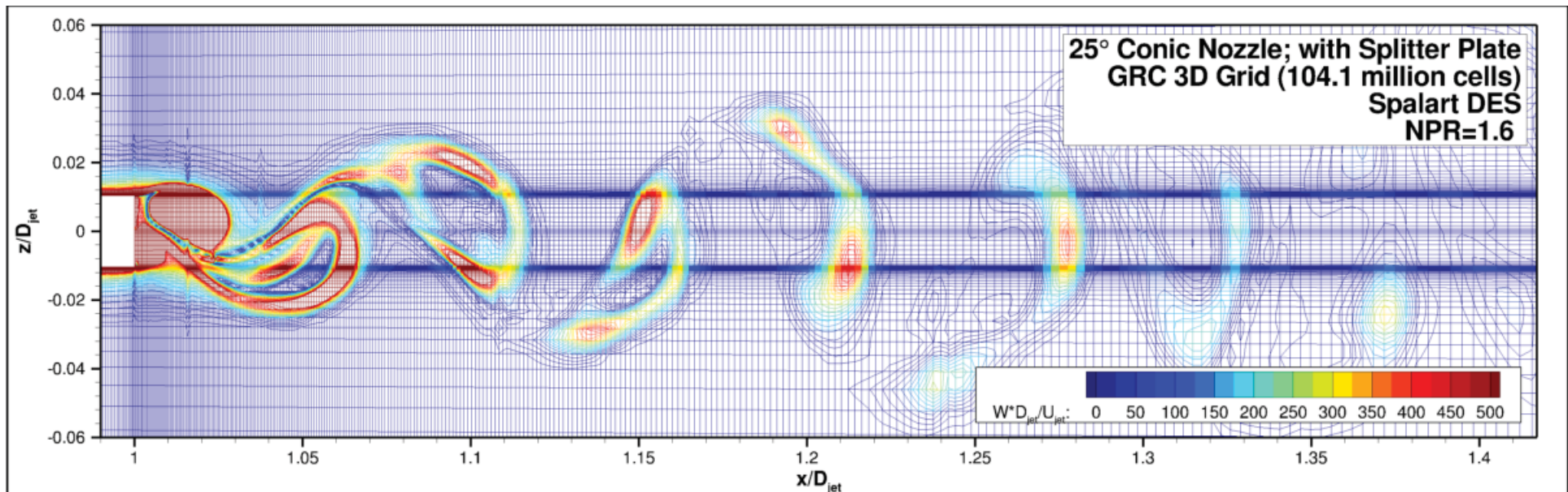
Instance 3: Comparison of Wind-US and Experimental Shadowgraphs



- Wind-US DES indicates a vortex shedding frequency of 28.7-31.2 kHz, somewhat lower than that reported at PAW 2012 (~32 kHz)
 - With Medium grid sequence, ~32 kHz vortex shedding frequency
- No unsteady behavior observed for PAW grid at NPR=1.6 with DES
 - Grid too dissipative?



Instance 3: Grid Resolution and Vorticity



- GRC Grid solution shown
 - PAW Grid solution: no unsteady flow evident
- Solutions assume flow is symmetric; only 180° sector modeled.
- Instantaneous solutions shown.
- Downstream mesh fine enough to resolving vortex structures.



Observations

- 10 of 22 axisymmetric solutions showed unsteadiness when running with constant CFL# and required constant time step to converge.
 - Included lower NPRs (≤ 2.5) and high NPRs (5.0, 7.0)
 - Included more GRC Grid solutions than PAW Grid solutions.
 - Convergence required 3-4 times as many iterations.
- Both grids are good for predicting nozzle performance (C_d and C_v)
 - Some difference between solutions for $\text{NPR} < 3.0$, no experimental data for clarity
- Both grids captured compression and expansion waves
 - Finer GRC grid was superior at resolving and pro-longing waves
- Curious why the provided 3D grid with splitter plate was configured with the symmetry plane aligned with the splitter plate.
 - This seemed a poor choice for observing unsteady vortex formation from splitter plate.
- Vortex shedding was only observed with DES using GRC grid
 - No evidence of unsteadiness using PAW grid



Wind-US Dual-Stream Nozzle Results for the AIAA 2nd Propulsion Aerodynamics Workshop

Vance Dippold, III
Mina R. Mankbadi
Lance Foster
NASA Glenn Research Center

July 31-August 1, 2014



Dual Separate Flow Reference (DSFR) Nozzle

- Ambient Conditions
 - $p_{amb}=14.24$ psi
 - $T_{amb}=520.0^{\circ}$ R
- Fan Stream Inflow
 - Inflow boundary (no profile) with no boundary layer
 - Fan Total Pressure: $p_{0,fan}/p_{amb}$ varies from 1.4-2.6
 - Fan Total Temperature: $T_{0,fan} = 530.0^{\circ}$ R for all cases
 - Turbulence Intensity = 5%;
ratio of turbulent to molecular viscosity = 1.0
- Core Stream Inflow
 - Core Total Pressure: $p_{0,core} = p_{0,fan} / 1.2$
 - Core Total Temperature: $T_{0,core} = 530.0^{\circ}$ R for all cases
 - Turbulence Intensity = 5%;
ratio of turbulent to molecular viscosity = 1.0



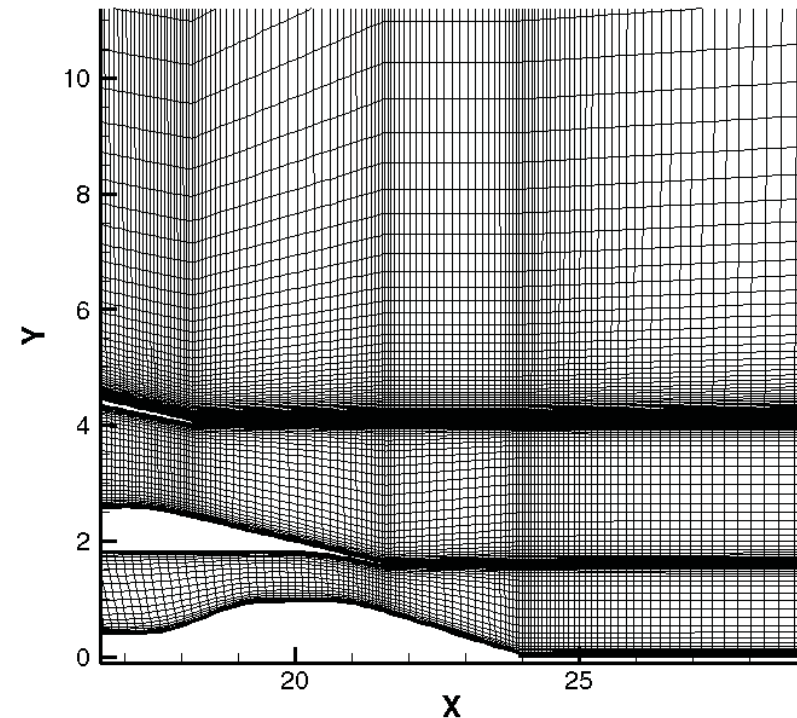
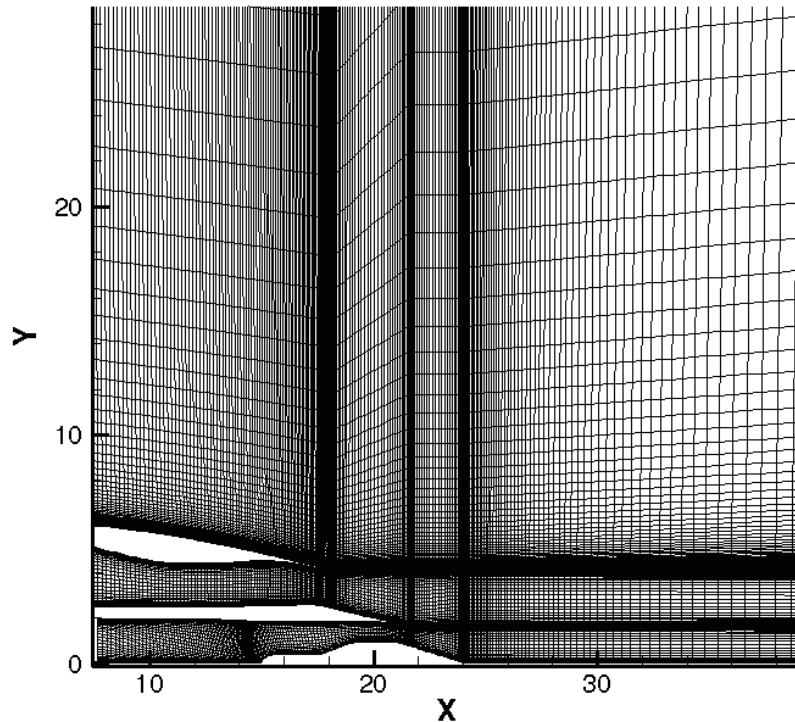
Analytical Tools

- Wind-US v3
 - RANS and Hybrid RANS/LES solvers
 - Structured and unstructured grids
 - Numerous turbulence models, numerical schemes, and boundary conditions
- All cases:
 - Structured grid solver
 - RANS with SST turbulence model (no compressibility corrections)
 - Roe 2nd-order physical spatial integration scheme (default)
 - Minmod TVD grid flux limiter (default)
 - Inflows: p_0 , T_0 held constant
 - Outflow: p_{amb} held constant



Grid

- 2-D Axisymmetric, structured, point-matched
- Created with Pointwise
- Based on PAW-supplied structured grids



- 86,259 grid points
- 11 zones
- $(\Delta s)_{wall} = 5e-5$ inches
- $(\Delta s)_{exit} = 5e-2$ inches



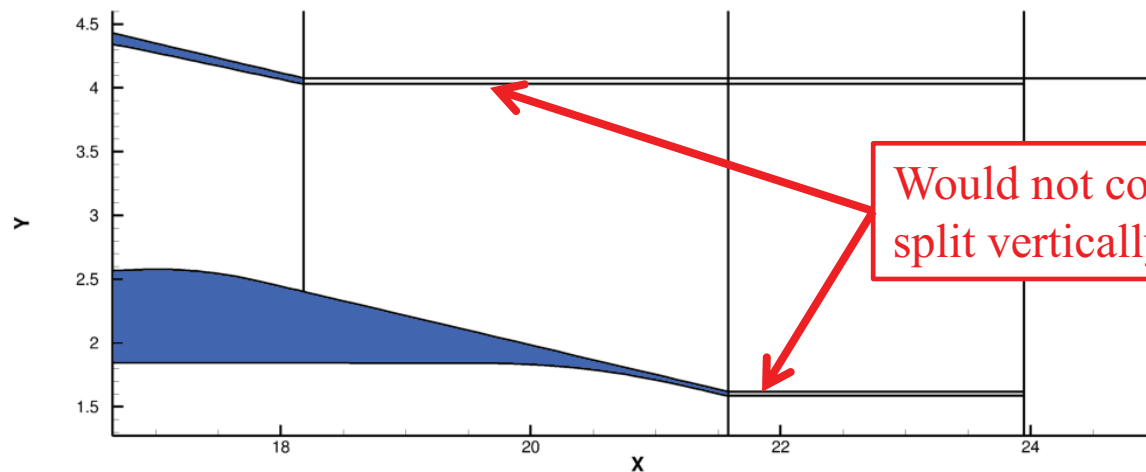
Convergence

- 5 iterations/cycle for all steady-state cases
- Coarse Mesh: 4,000 iterations
- Medium Grid: 10,000 iterations
- Fine Grid: time-accurate
 - 20,000 iterations (if initially run time-accurate)
 - 160,000 iterations (if initially run steady-state)
- Grid zones had to be split and reassembled for convergence

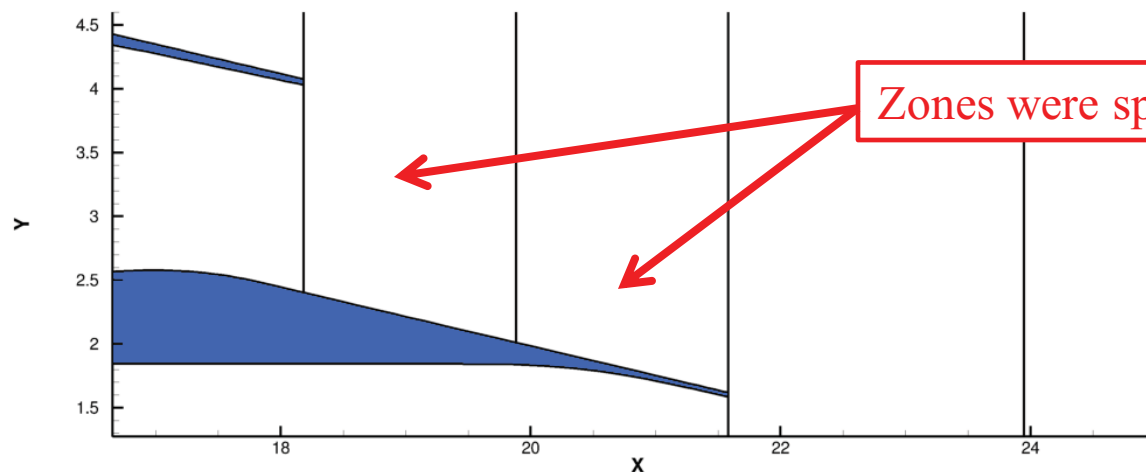


Grid Topology

AIAA



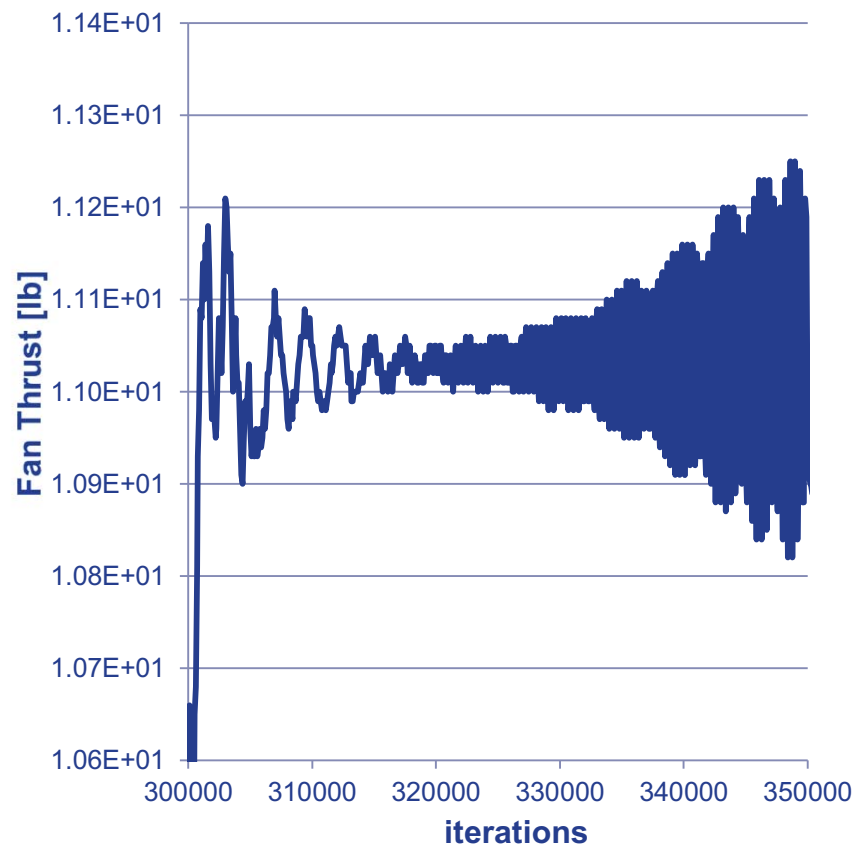
GRC



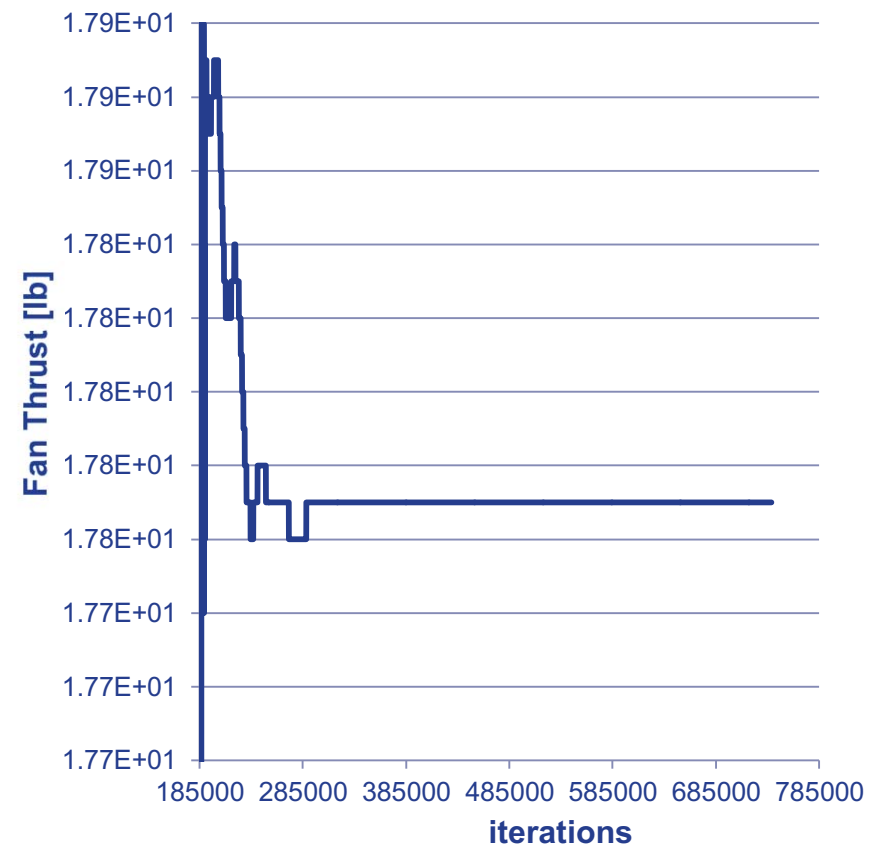


Fine Mesh Divergence in Steady-State

Steady-State (CFL#=0.25)



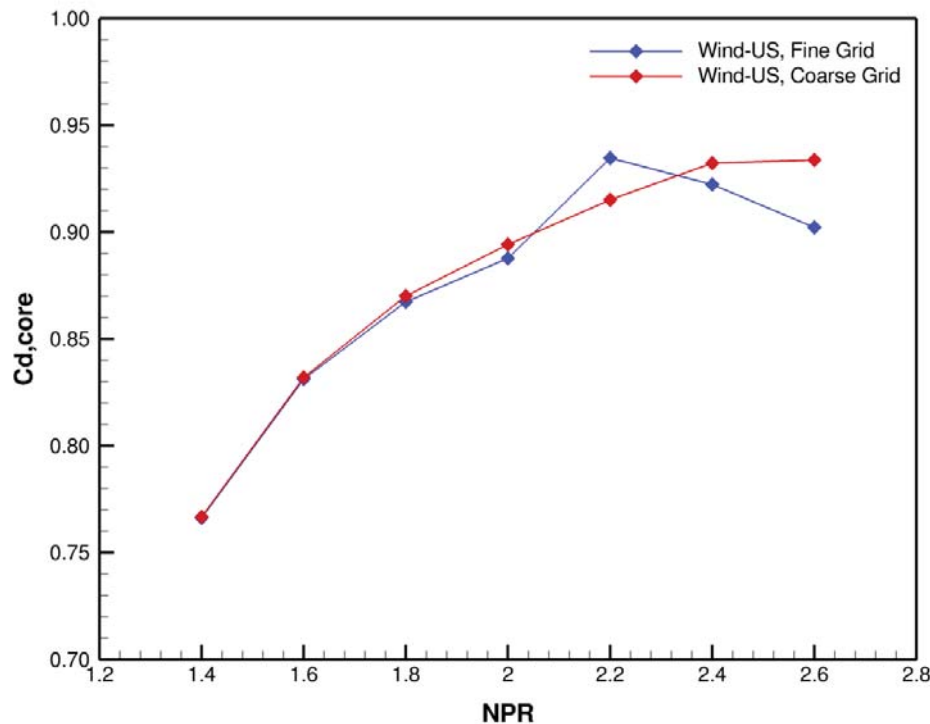
Time-accurate ($\Delta t=5e-9$)



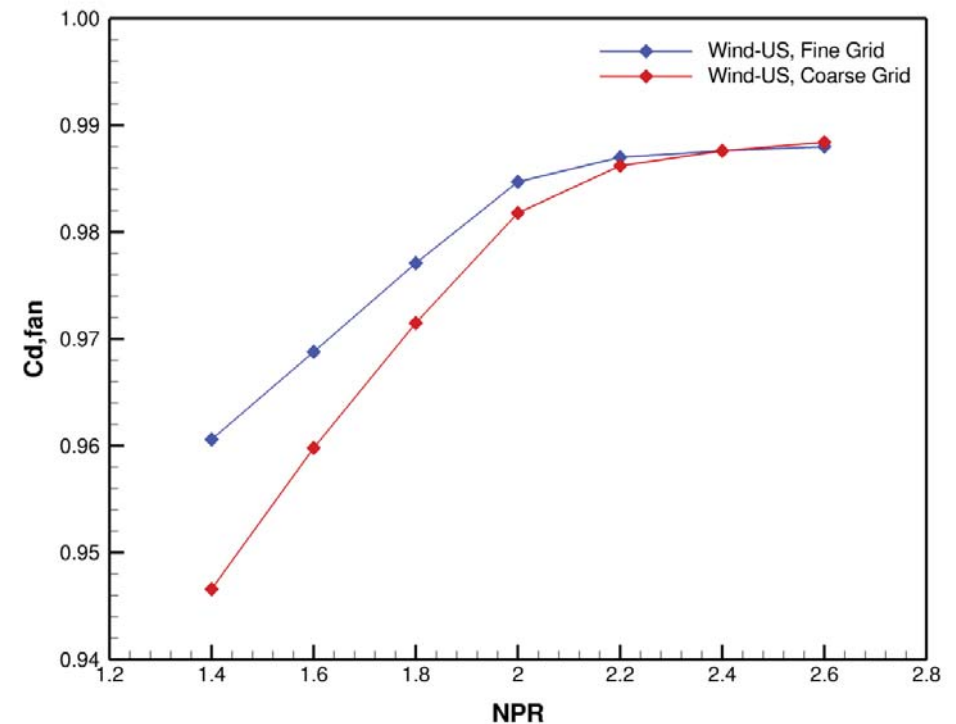


Fan and Core Stream Discharge Coefficients

Core Stream



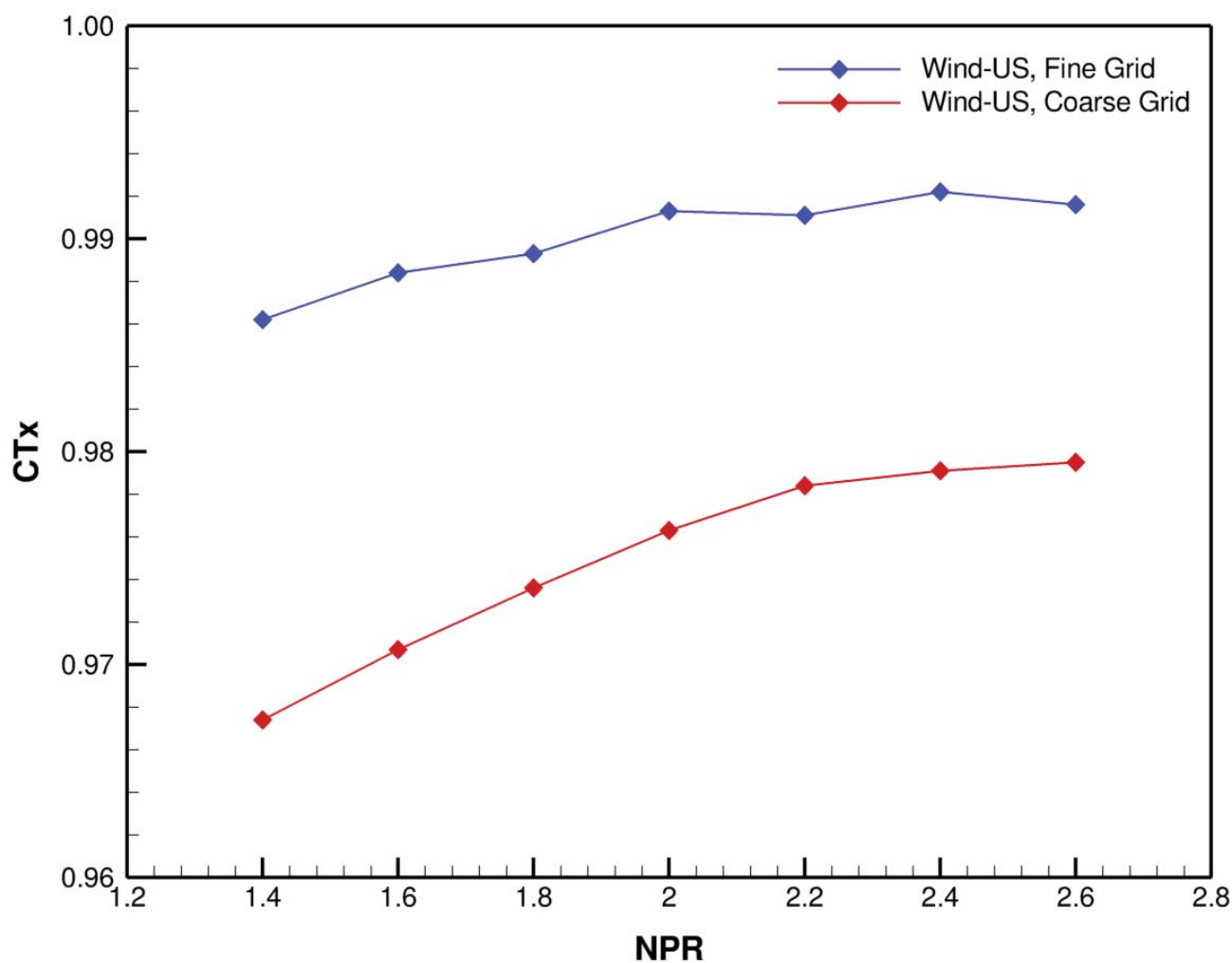
Fan Stream



Note: NPR is Fan Stream NPR ($p_{0,fan}/p_{amb}$)



Total Thrust Coefficient

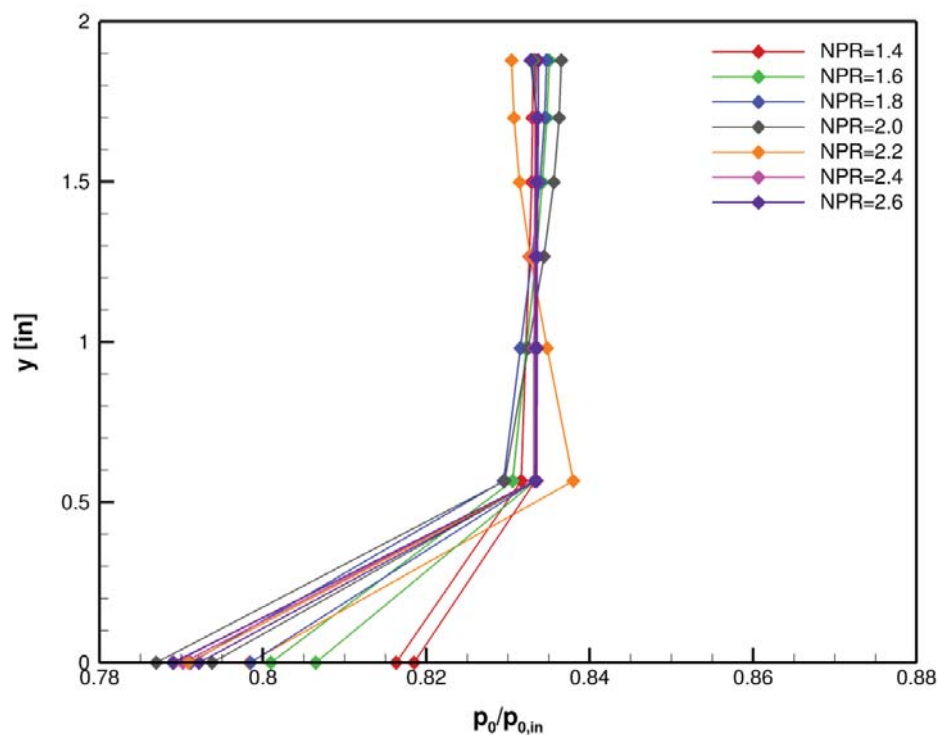


Note: NPR is Fan Stream NPR ($p_{0,fan}/p_{amb}$)

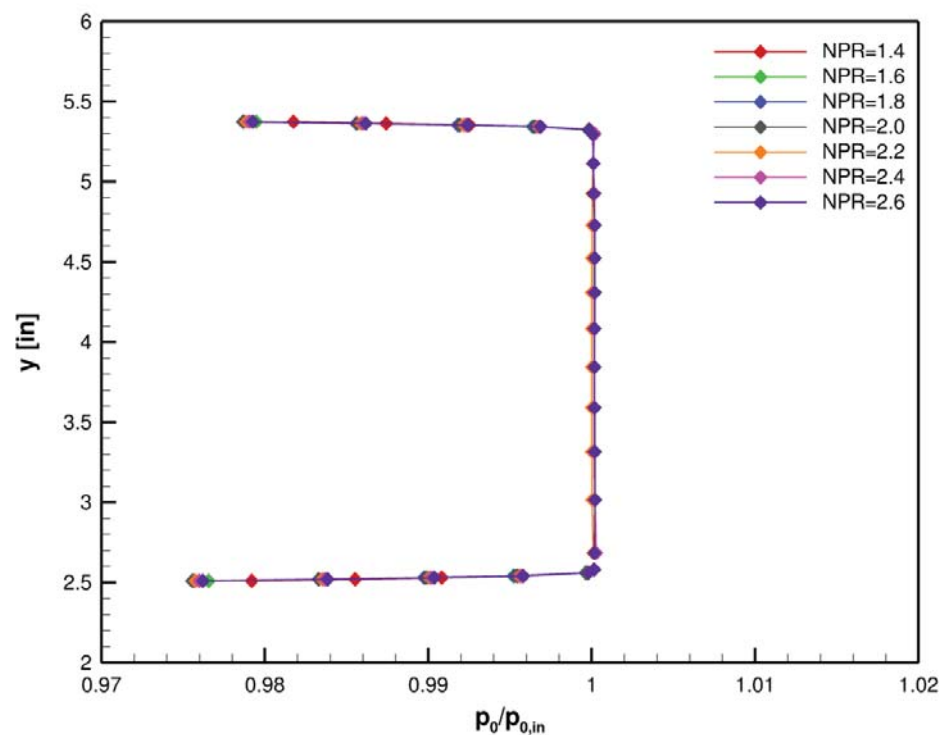


Fan and Core Stream Total Pressure Rakes

Core Stream



Fan Stream

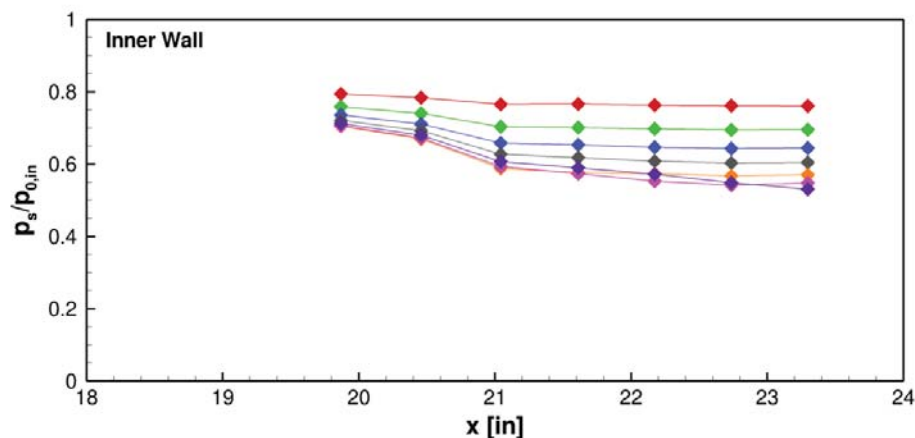
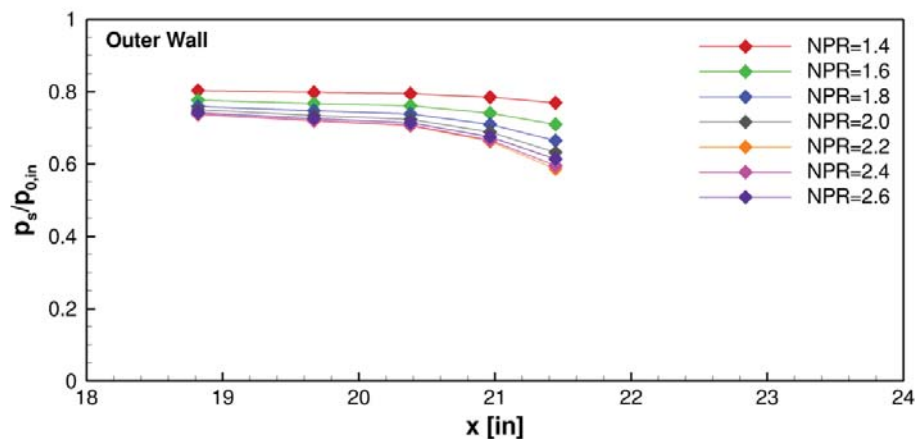


Note: NPR is Fan Stream NPR ($p_{0,fan}/p_{amb}$)

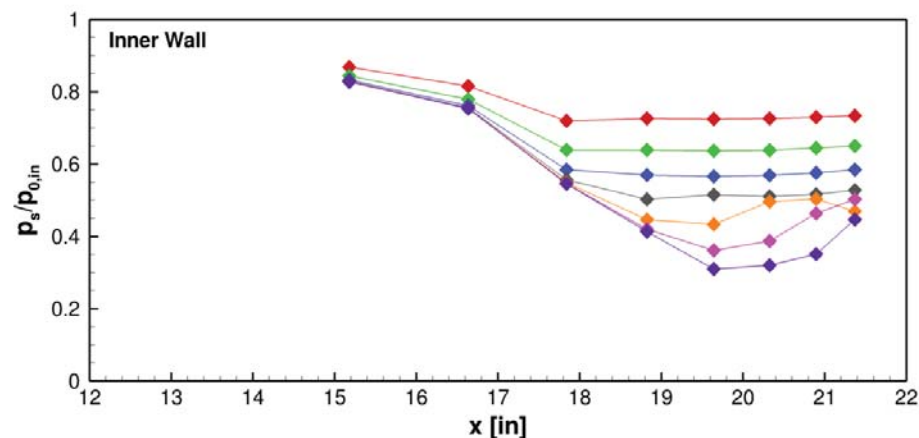
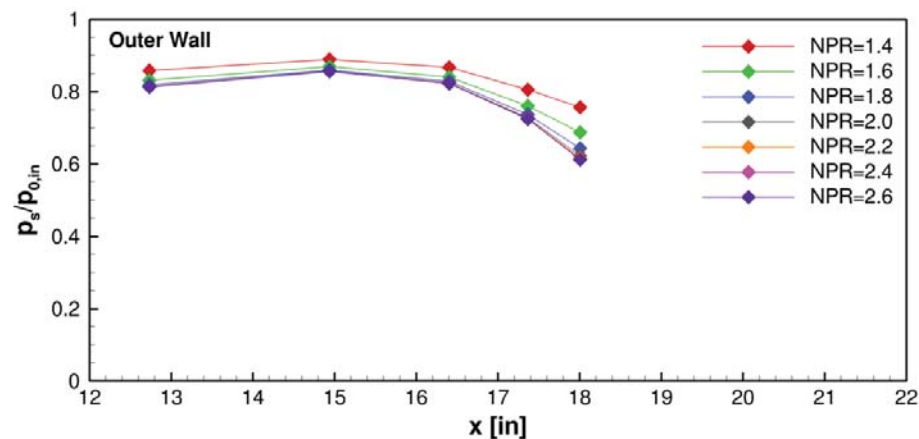


Fan and Core Stream Static Pressure Taps

Core Stream



Fan Stream



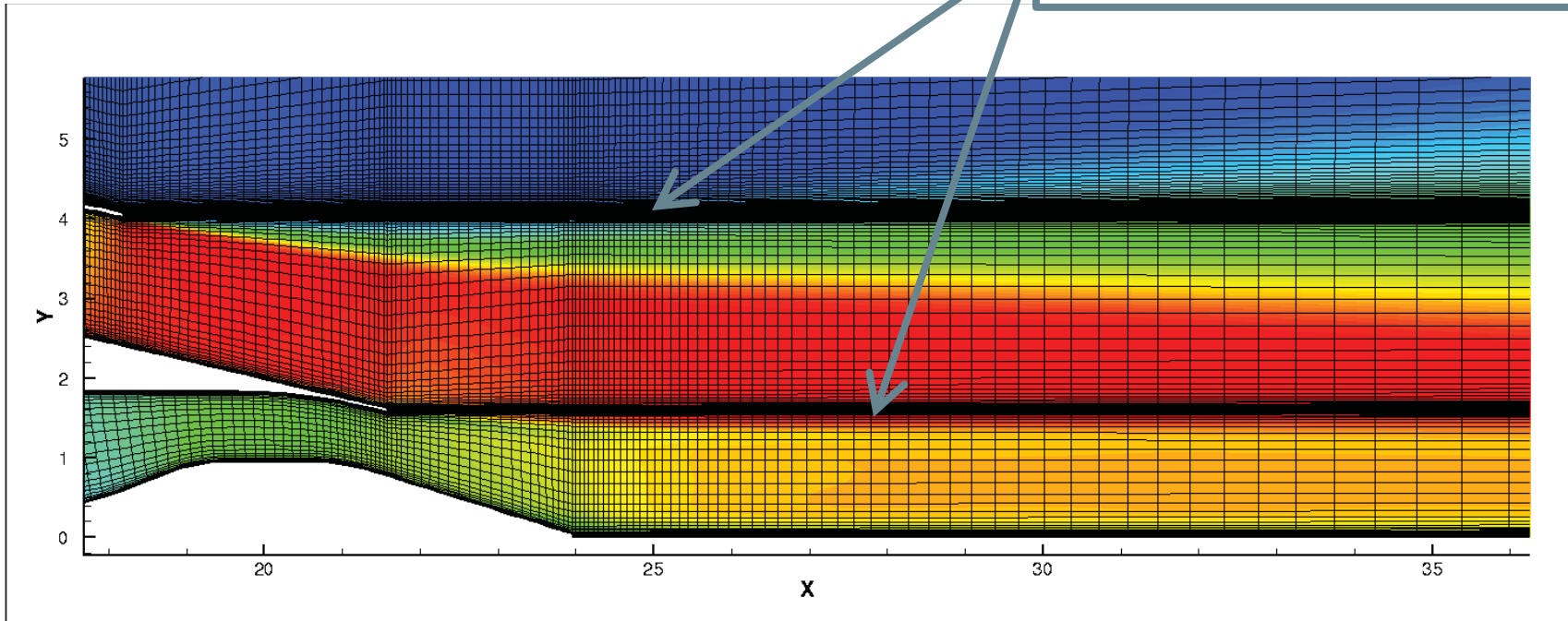
Note: NPR is Fan Stream NPR ($p_{0,fan}/p_{amb}$)



Mixing

Contours: Mach Number

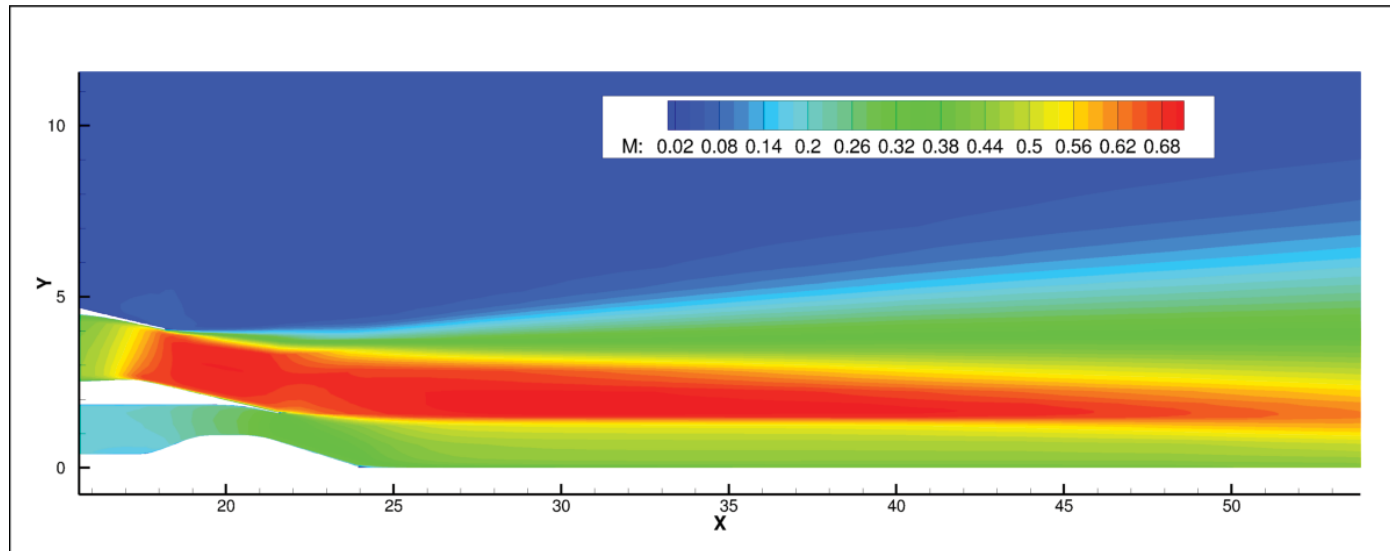
$$p_{0, \text{far}} / p_{\text{amb}} = 1.8$$



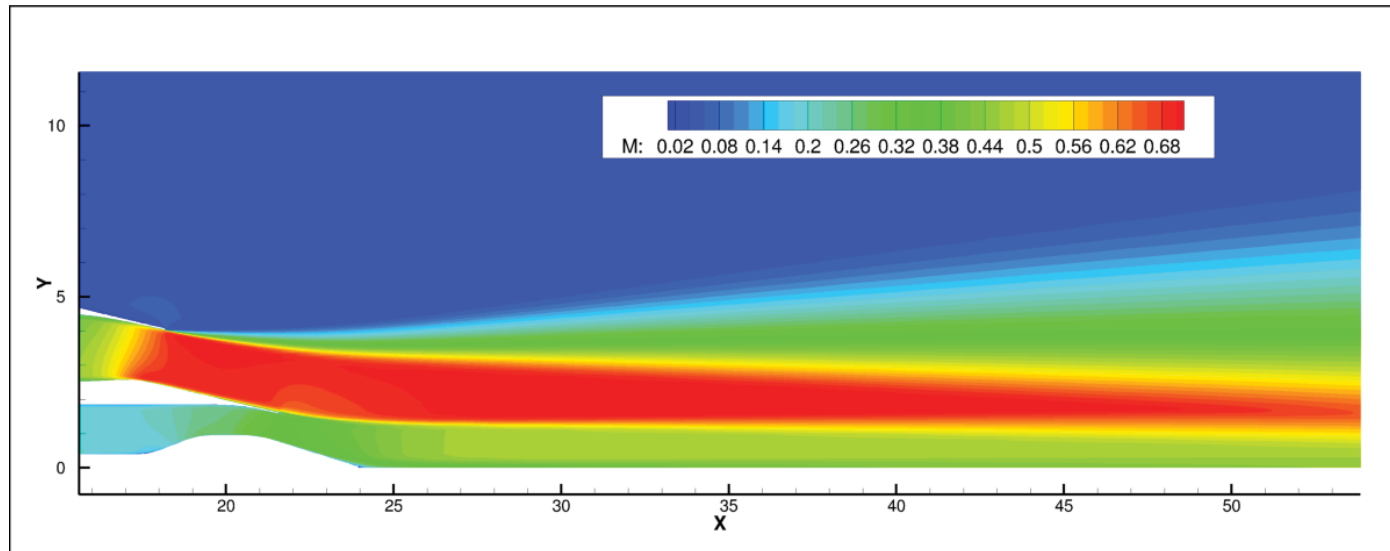


$$p_{0,fan}/p_{amb} = 1.4$$

Coarse Grid
Solution



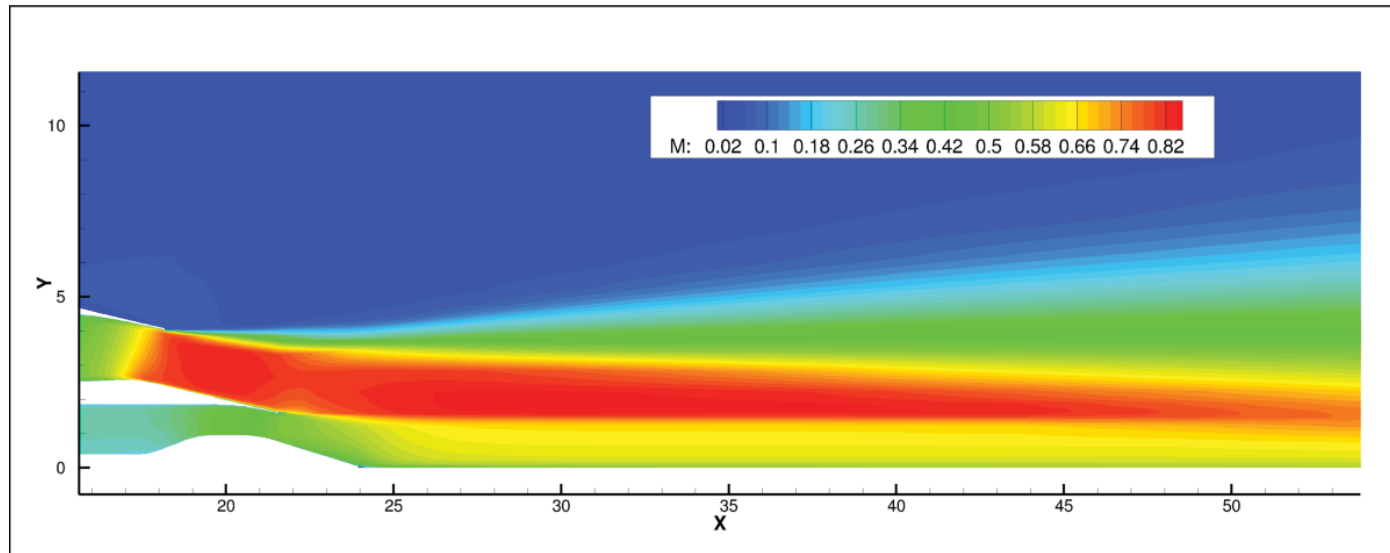
Fine Grid
Solution



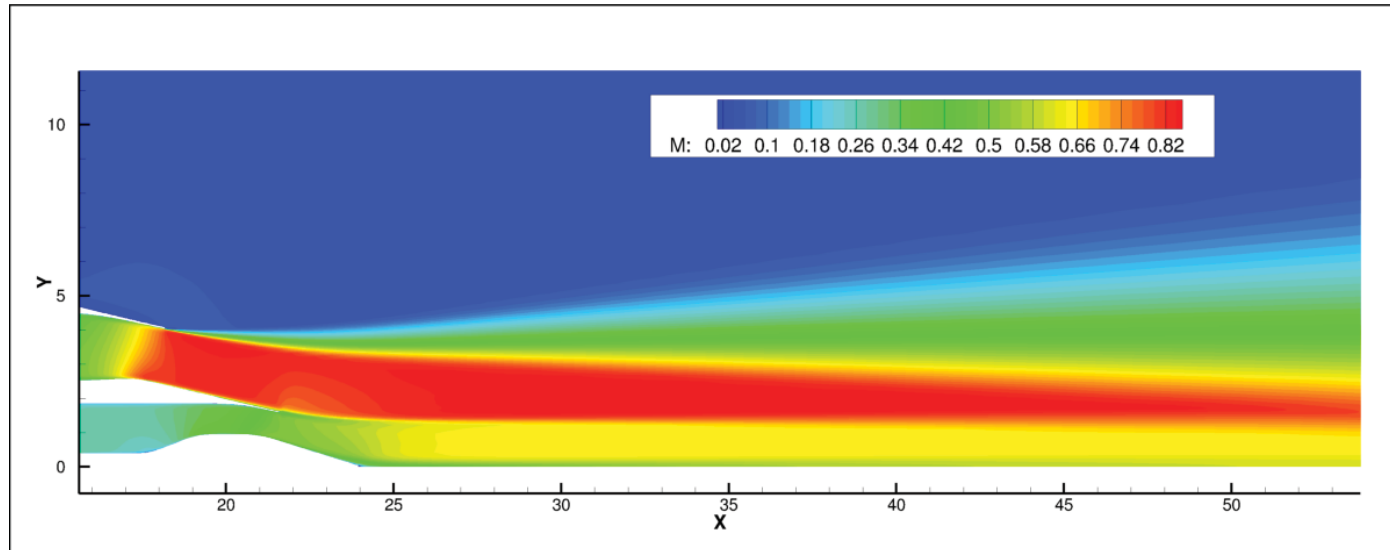


$$p_{0,fan}/p_{amb} = 1.6$$

Coarse Grid
Solution



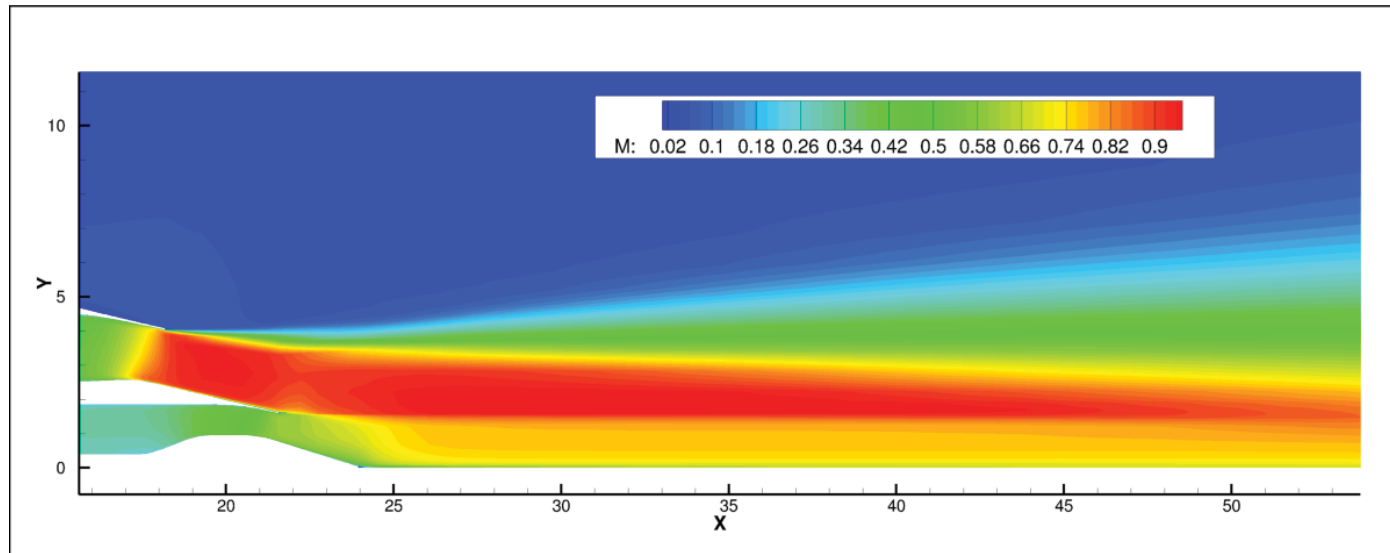
Fine Grid
Solution



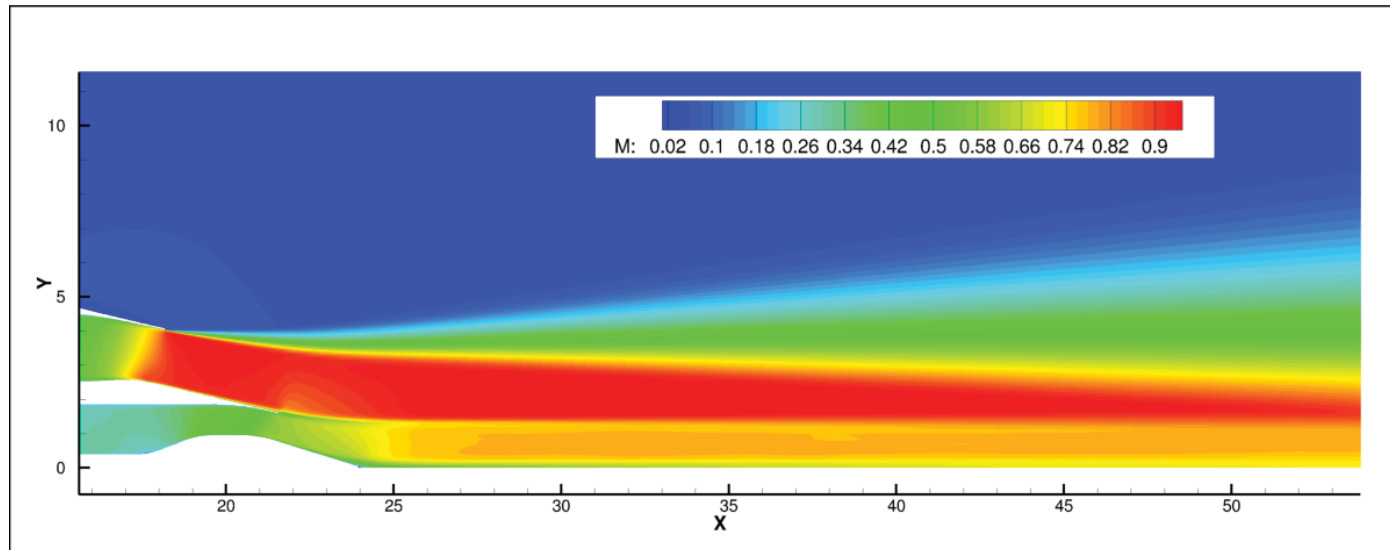


$$p_{0,fan}/p_{amb} = 1.8$$

Coarse Grid
Solution



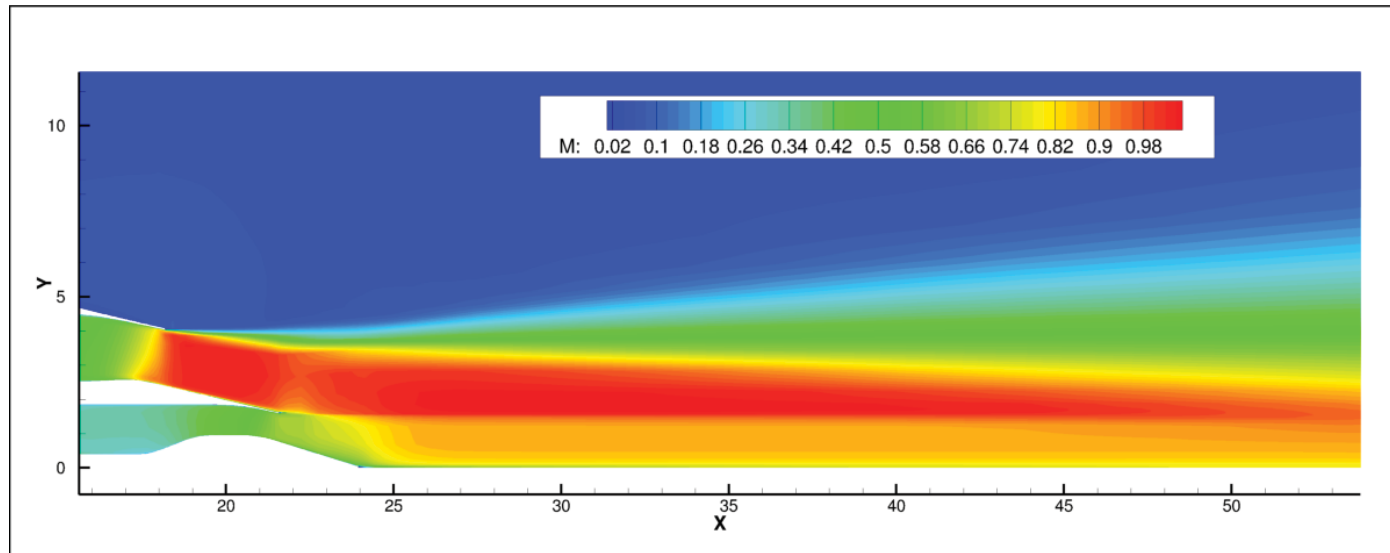
Fine Grid
Solution



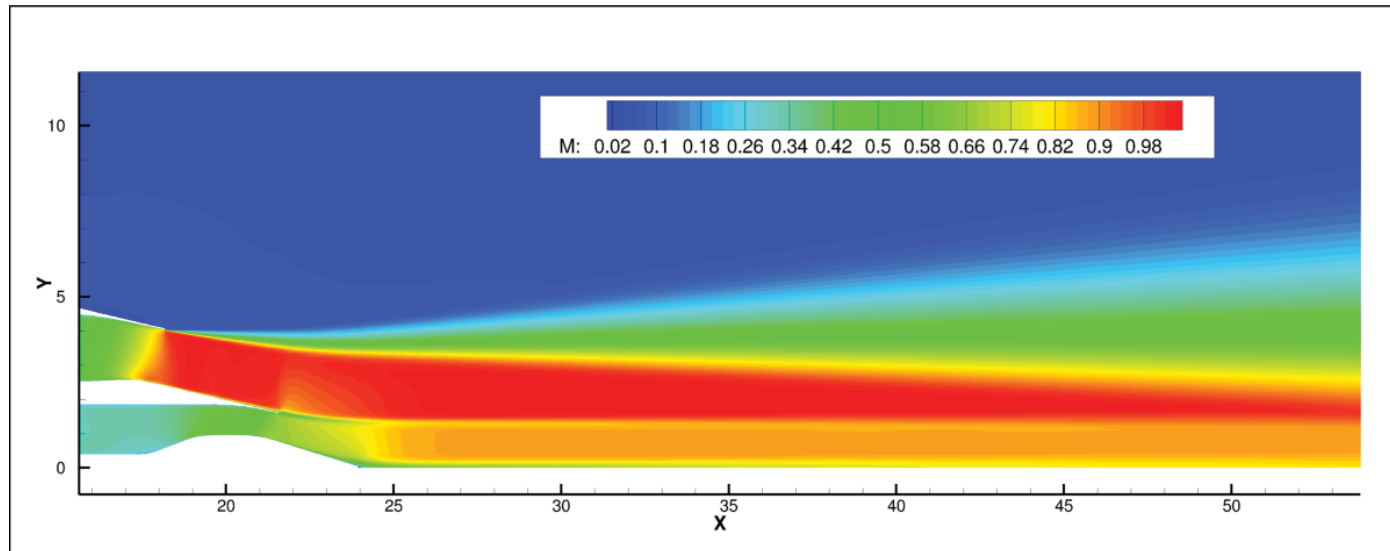


$$p_{0,fan}/p_{amb} = 2.0$$

Coarse Grid
Solution



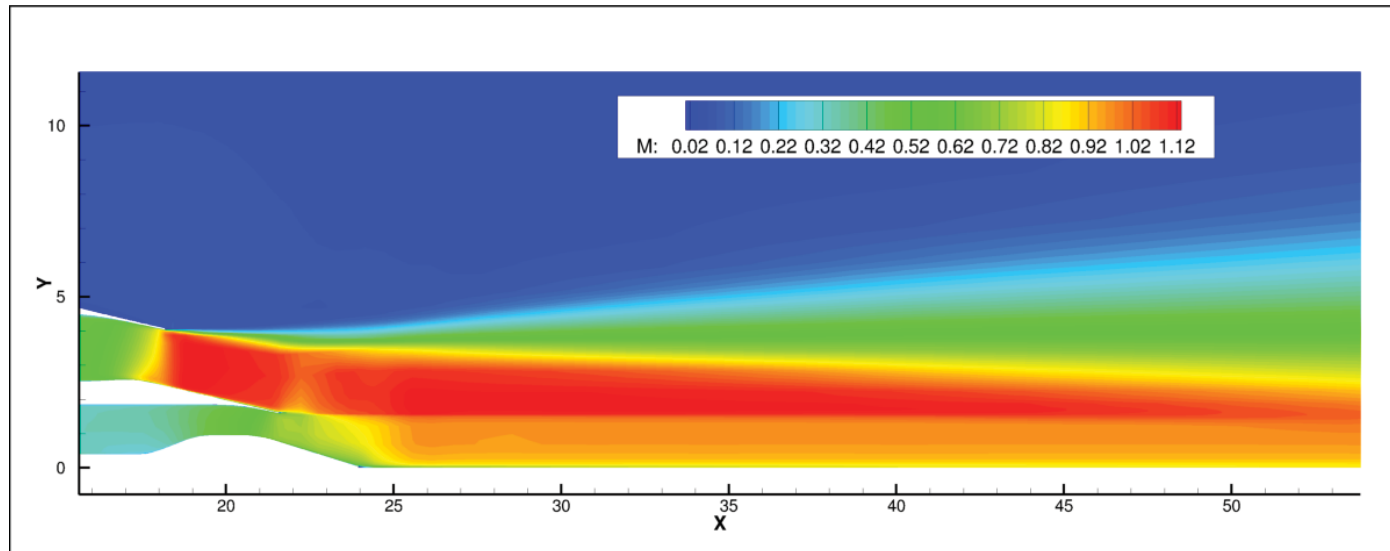
Fine Grid
Solution



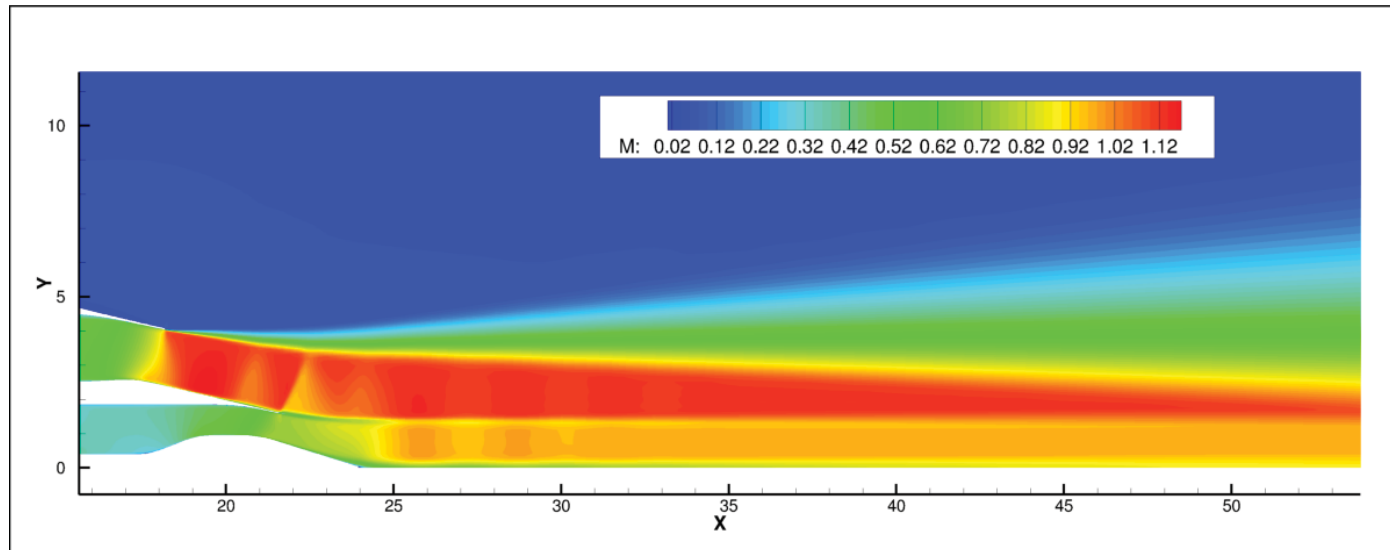


$$p_{0,fan}/p_{amb} = 2.2$$

Coarse Grid
Solution



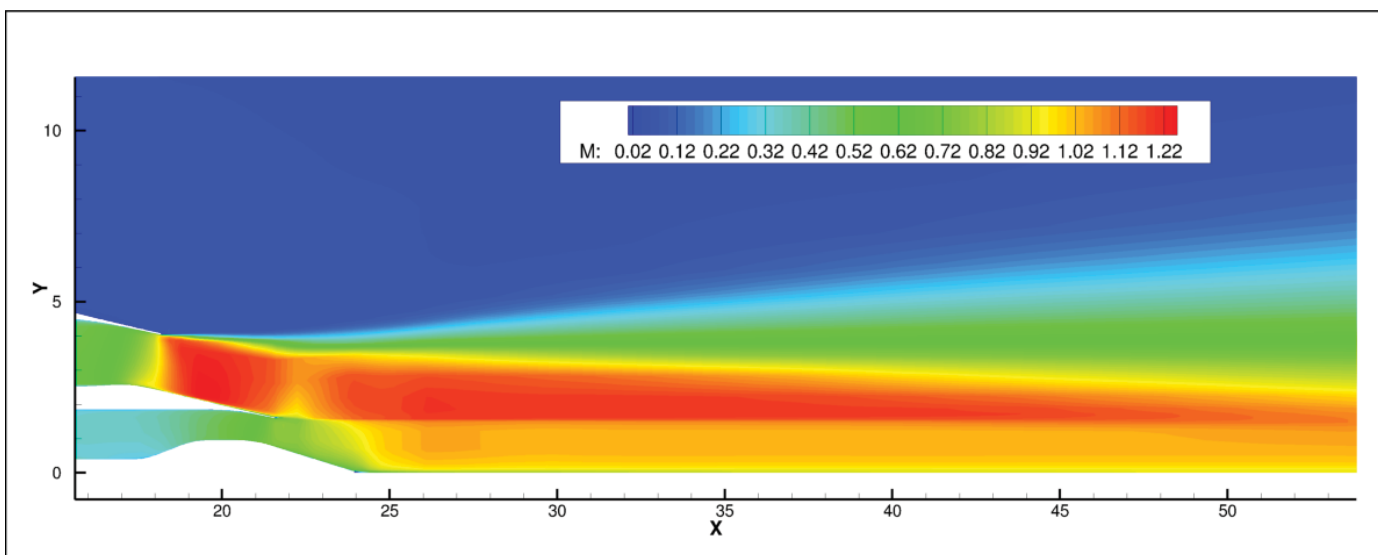
Fine Grid
Solution



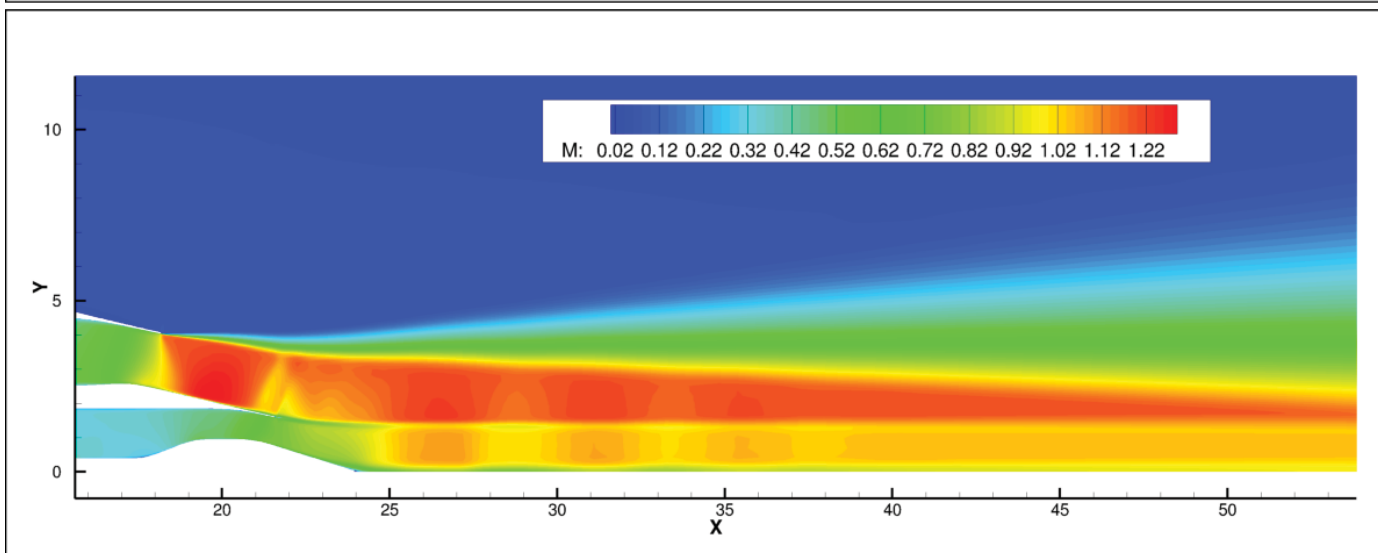


$$p_{0,fan}/p_{amb} = 2.4$$

Coarse Grid
Solution



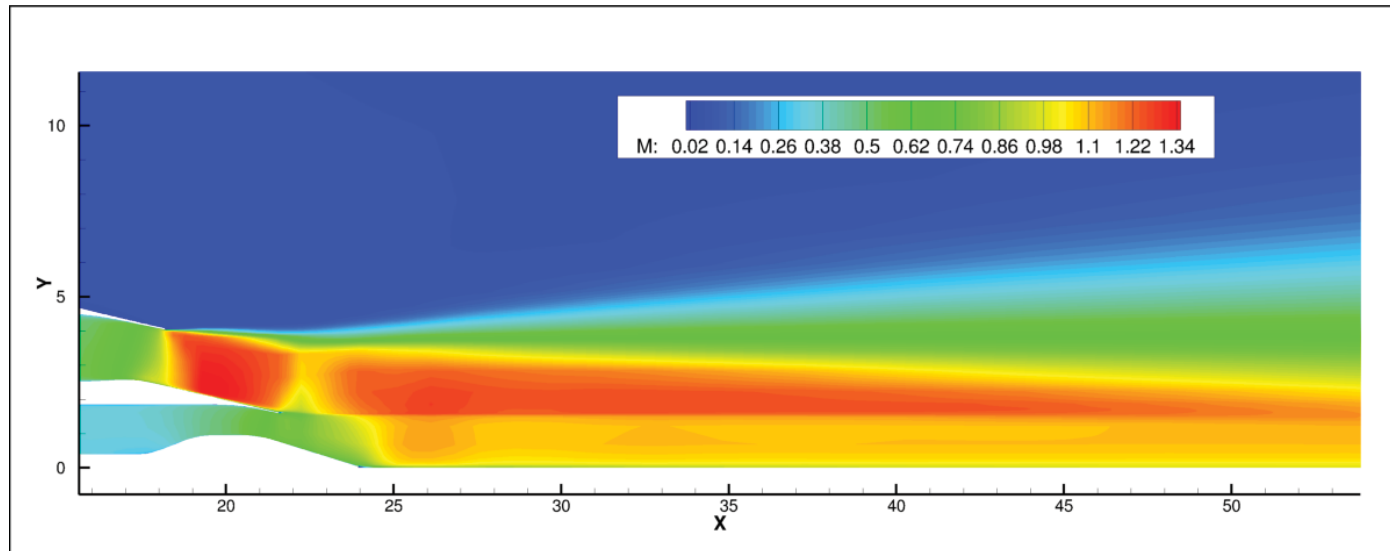
Fine Grid
Solution



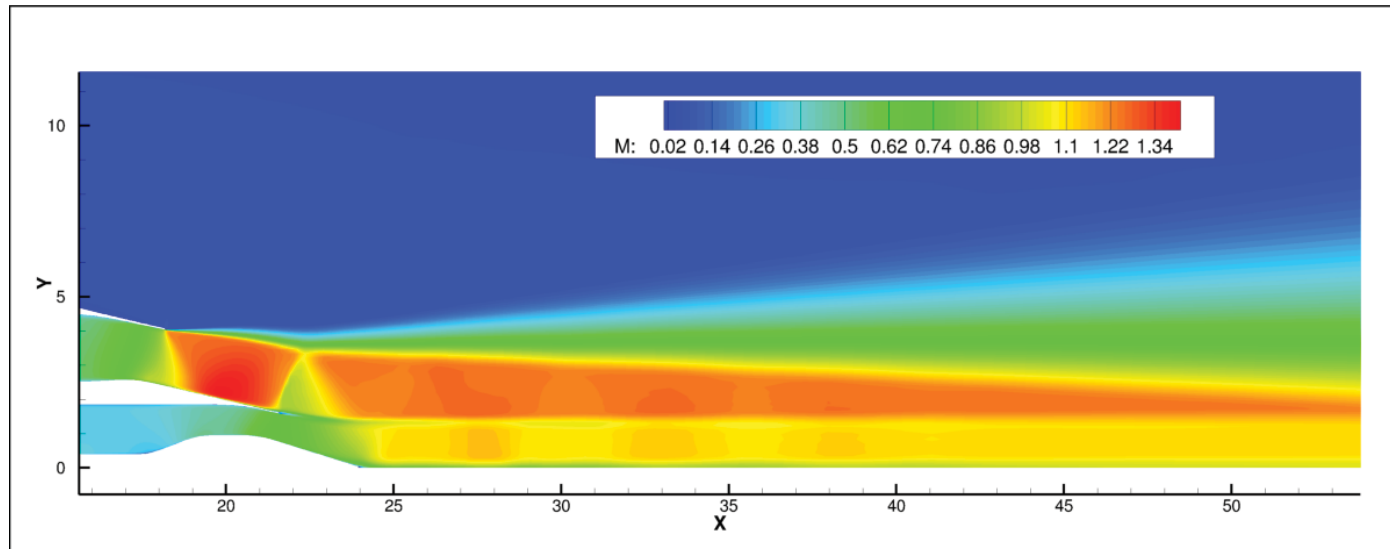


$$p_{0,fan}/p_{amb} = 2.6$$

Coarse Grid
Solution



Fine Grid
Solution





Wind-US S-Duct Results for the AIAA 2nd Propulsion Aerodynamics Workshop

Vance Dippold, III
Mina R. Mankbadi
Lance Foster
NASA Glenn Research Center

July 31-August 1, 2014



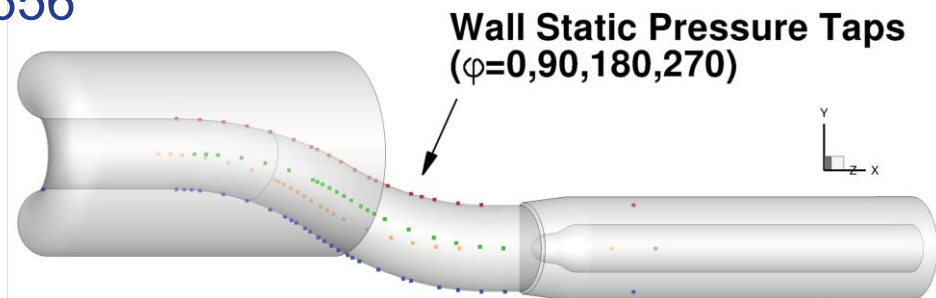
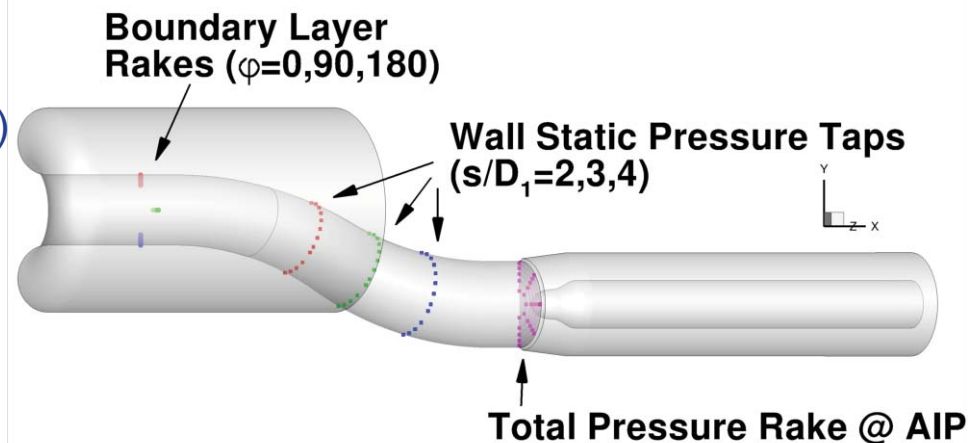
S-Duct Problem Description

- Geometry

- $D_1 = 133.15$ mm (at BL)
- $D_2 = 164.00$ mm (at AIP)
- Area Ratio = 1.52
- Length = $5.23 * D_1$
- Offset = $1.34 * D_1$

- Flow Conditions

- Tested in the R4MA facility at ONERA in 2006.
Run 1112, Data Point 656
- $p_0 = 88,744$ Pa
- $T_0 = 286.2$ K
- massflow = 2.427 kg/s
(for full 360°)
- AIP Mach = 0.3549





Wind-US

- Wind-US Version 3
- Used Grid provided by PAW
 - 20 Million points
 - Split into 385 Zones to increase multiprocessor efficiency and reduce computational time.
 - *Efficiency*: 90 iterations per min (10,000 iterations /110 minute)
- Use symmetry: only model half of the geometry
- Inflow: Specified p_0 , T_0 , Mach 0.01
- Outflow: Specified mass flow 2.427 kg/s * 0.5 (symmetry)
- External Outflow: p_0 12.8712, Mach 0.01
- Turbulence models
 - Menter Shear Stress Transport (SST)
 - Spalart-Allmaras (S-A)
- Full description of code features:
 - <http://www.grc.nasa.gov/WWW/wind/index.html>



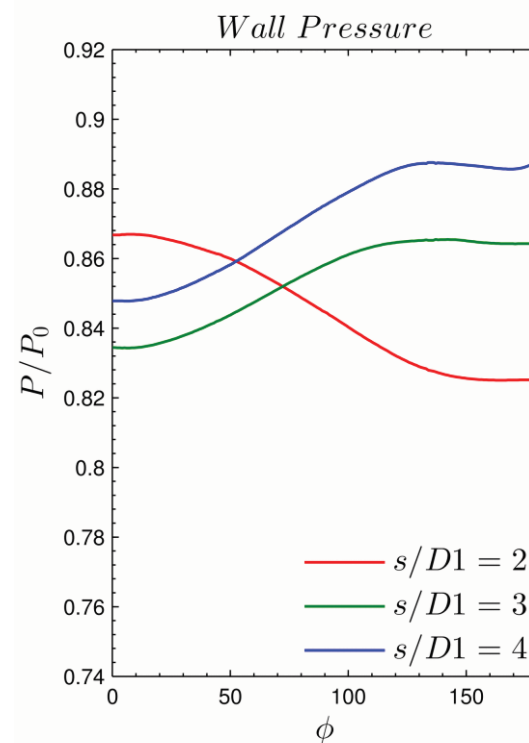
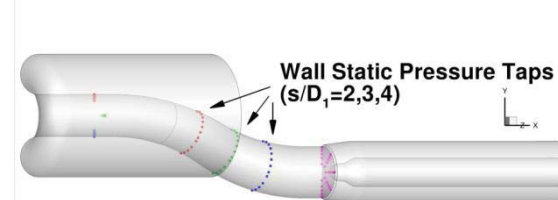
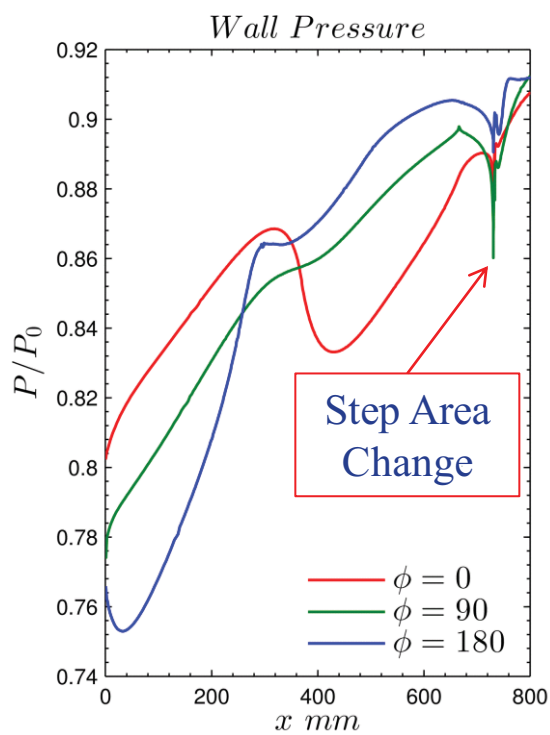
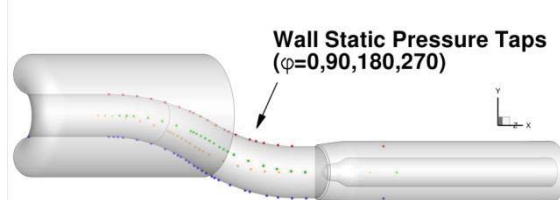
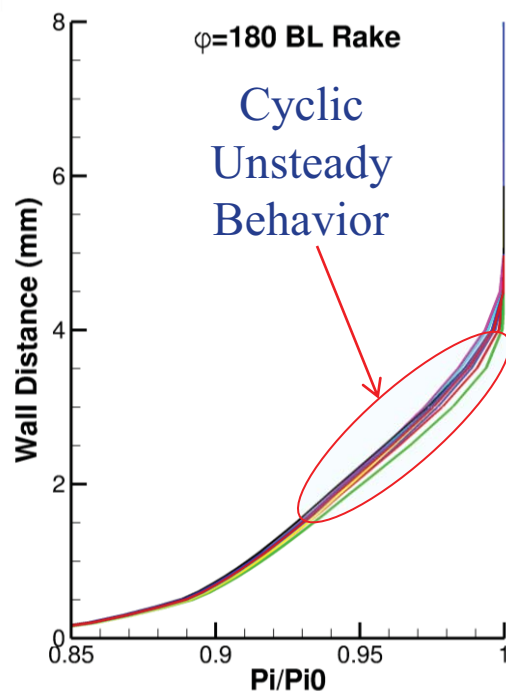
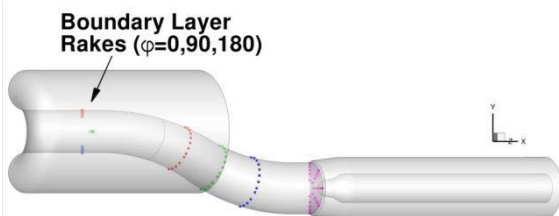
Solution Convergence

- Convergence to “steady-state”.
 - Solutions were shown to be periodically unsteady.
- Constant CFL of 0.5
- Ran for 600,000 iterations to eliminate transience and show unsteady behavior
- Last 100,000 iterations was looked at to evaluate convergence (plotted Boundary Layer every 10,000 iterations)
- Typical runtime on NASA Pleiades is 124 minutes per 10,000 iterations (wall time)



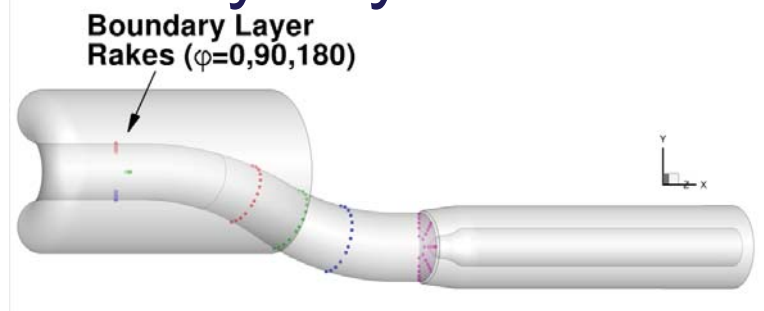
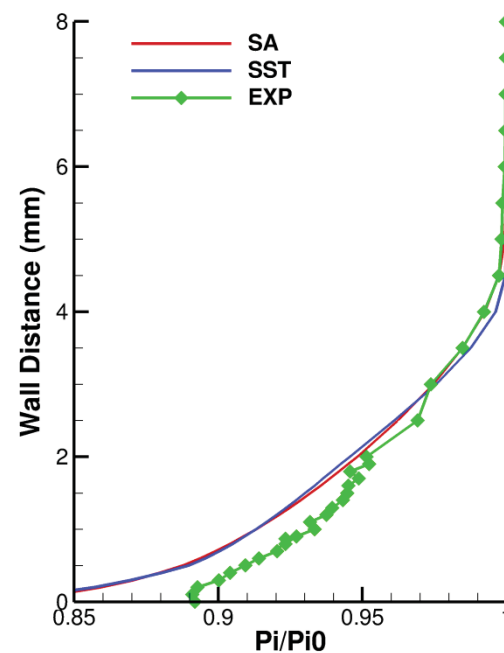
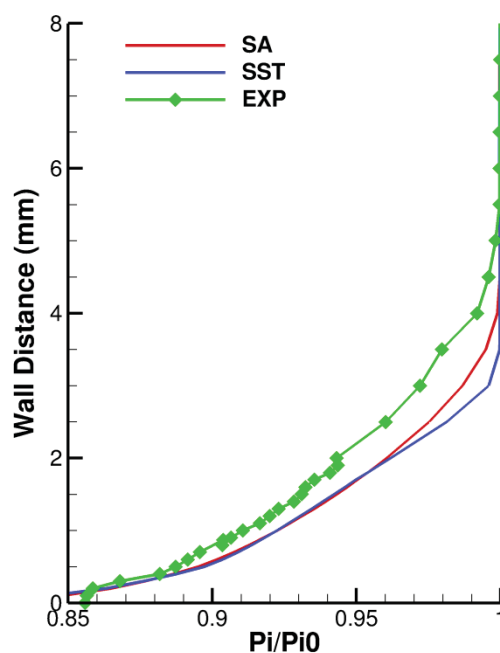
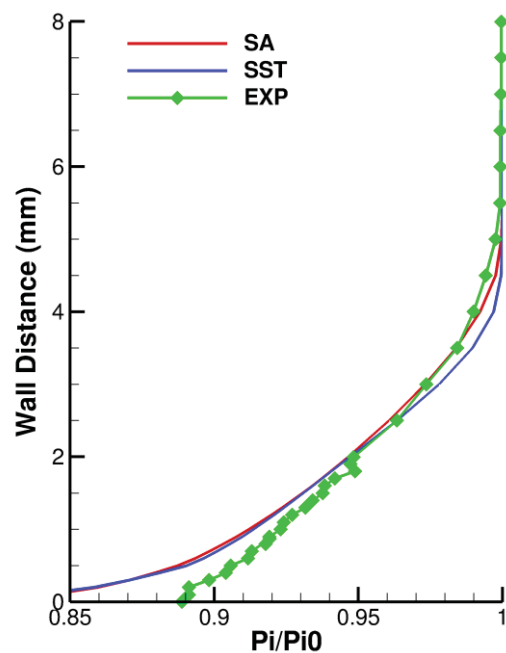
Solution Convergence on the Fine Grid

SST



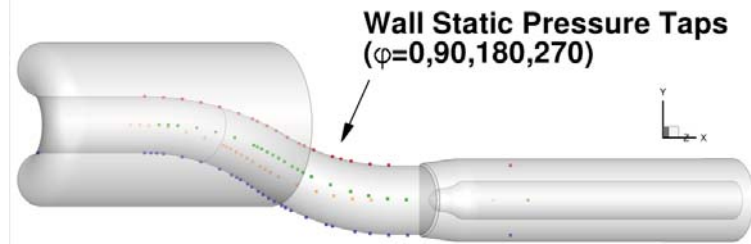
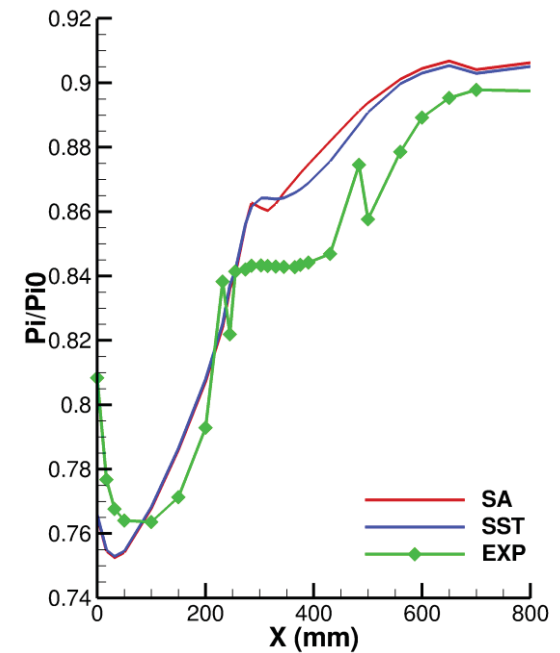
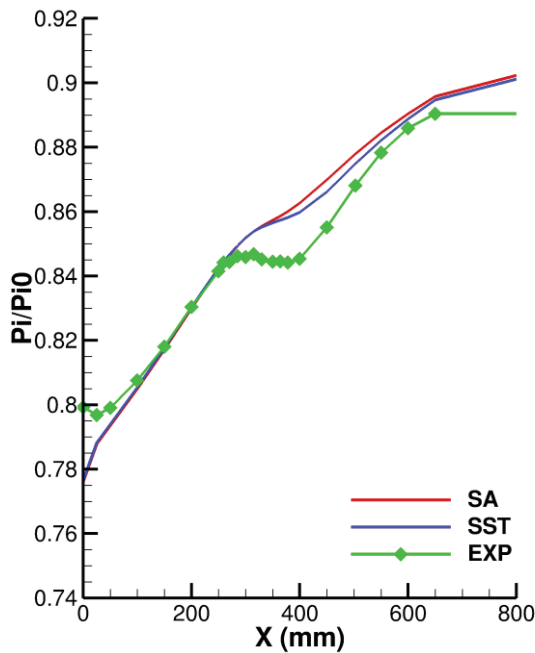
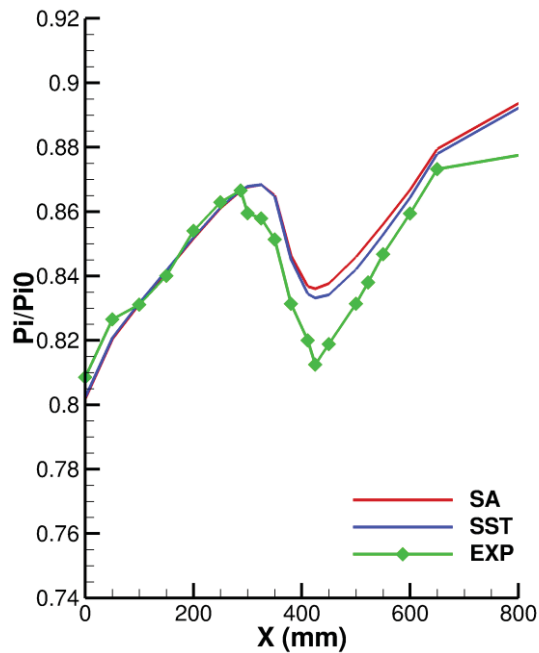


Boundary Layer Rake Data

 $\Phi=0^\circ$ $\Phi=90^\circ$ $\Phi=180^\circ$ 

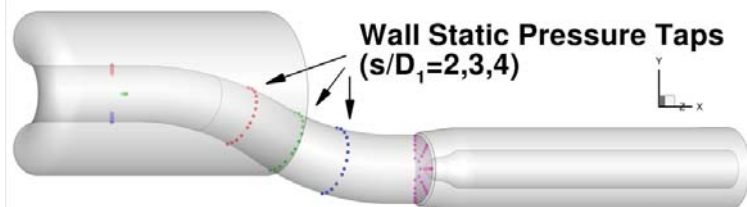


Streamwise Pressure Variation

 $\Phi=0^\circ$ $\Phi=90^\circ$ $\Phi=180^\circ$ 



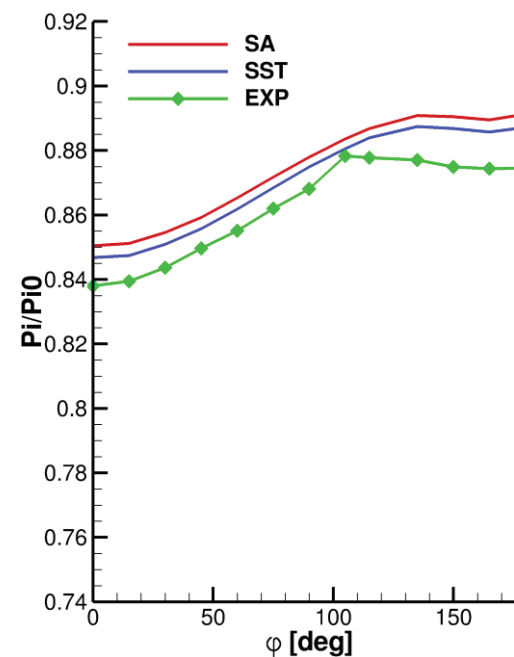
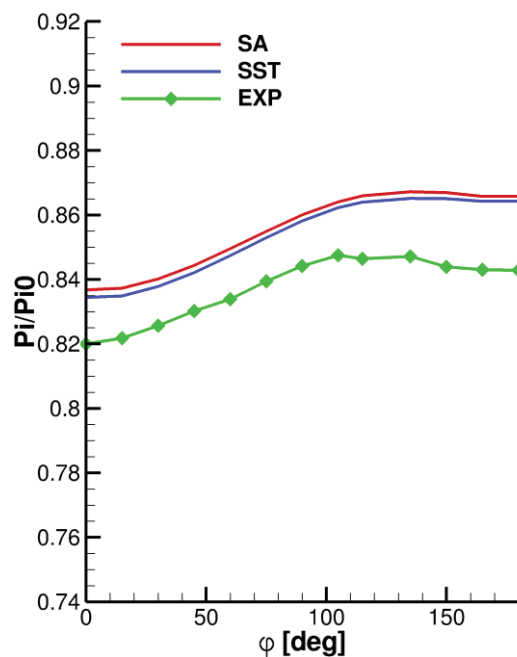
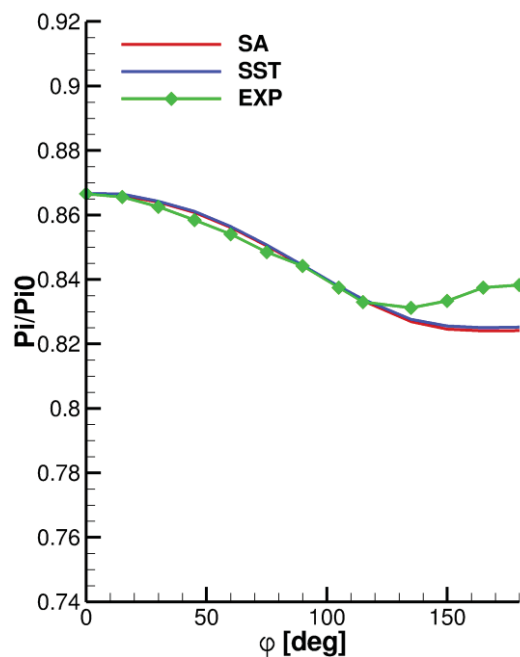
Circumferential Pressure Variation



$s/D_1=2$

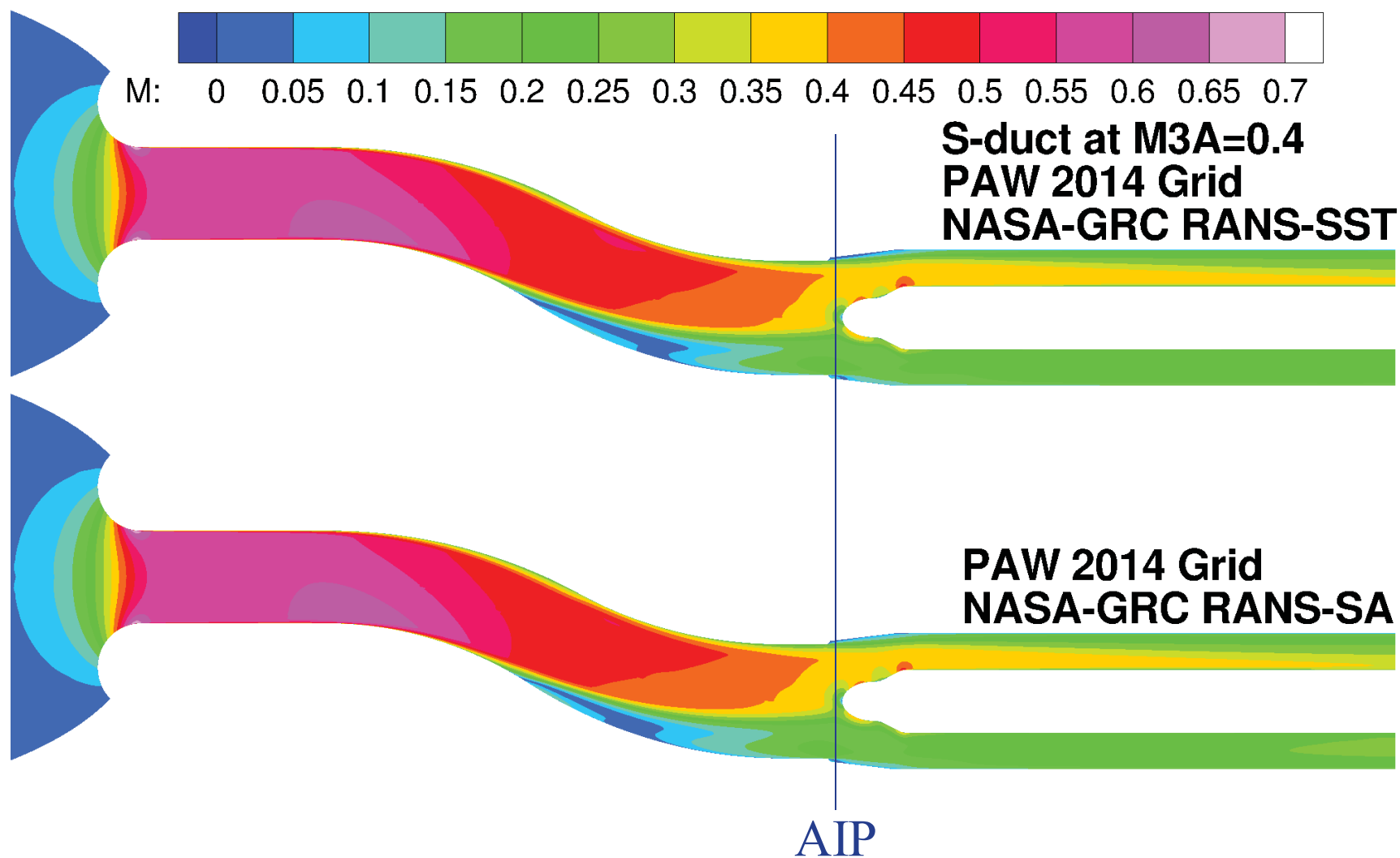
$s/D_1=3$

$s/D_1=4$





Symmetry Plane – Fine Grid

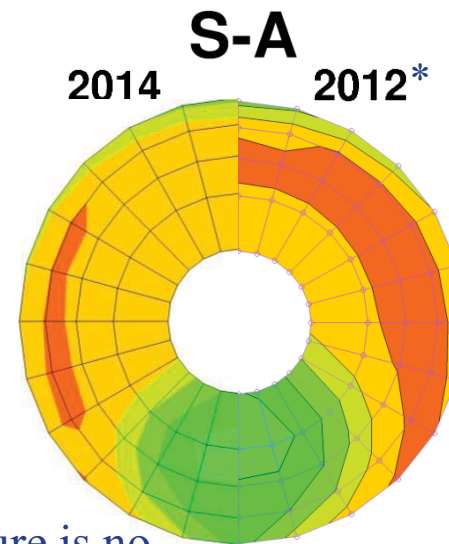
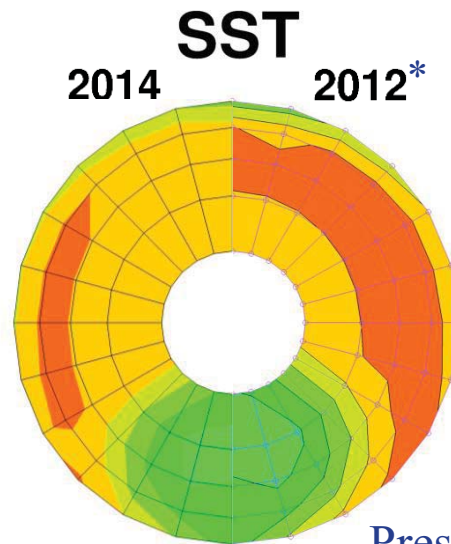
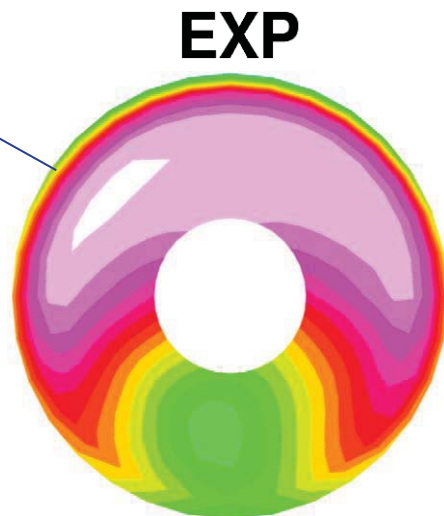




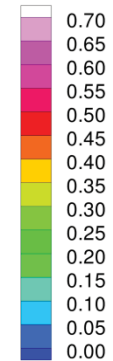
AIP Virtual Total Pressure Rake

P0 Recovery: SST=0.9771
SA=0.9777

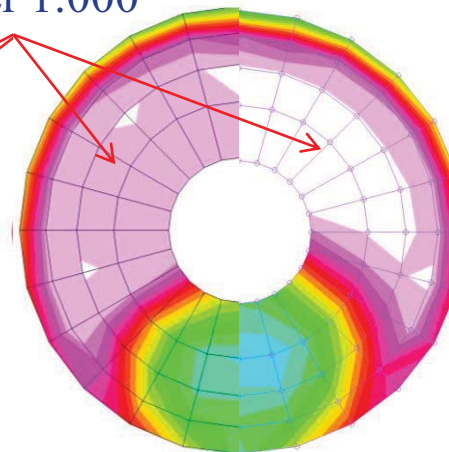
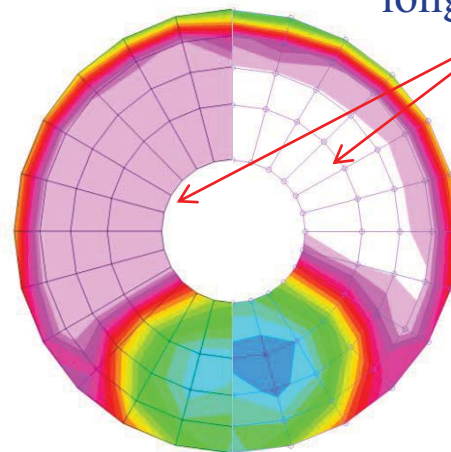
Note the asymmetry
in the experiment,
the computations are
that of a half section.



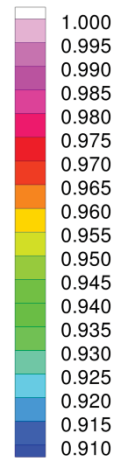
Mach



Pressure is no
longer 1.000



Pi/Pi0



*2012 CFD results were provided by Dennis Yoder.



Observations

- SST is slightly unsteady while SA is still unsteady but to a lesser degree.
- Incoming boundary layer profiles match well with experimental data.
- There is no significant difference between SA and SST predictions when compared with the experiment.
- 2014 simulations are in good agreement with 2012 simulations except the 2014 p_o recovery is slightly higher.
- The distortion values of 2014 and 2012 are not the same.